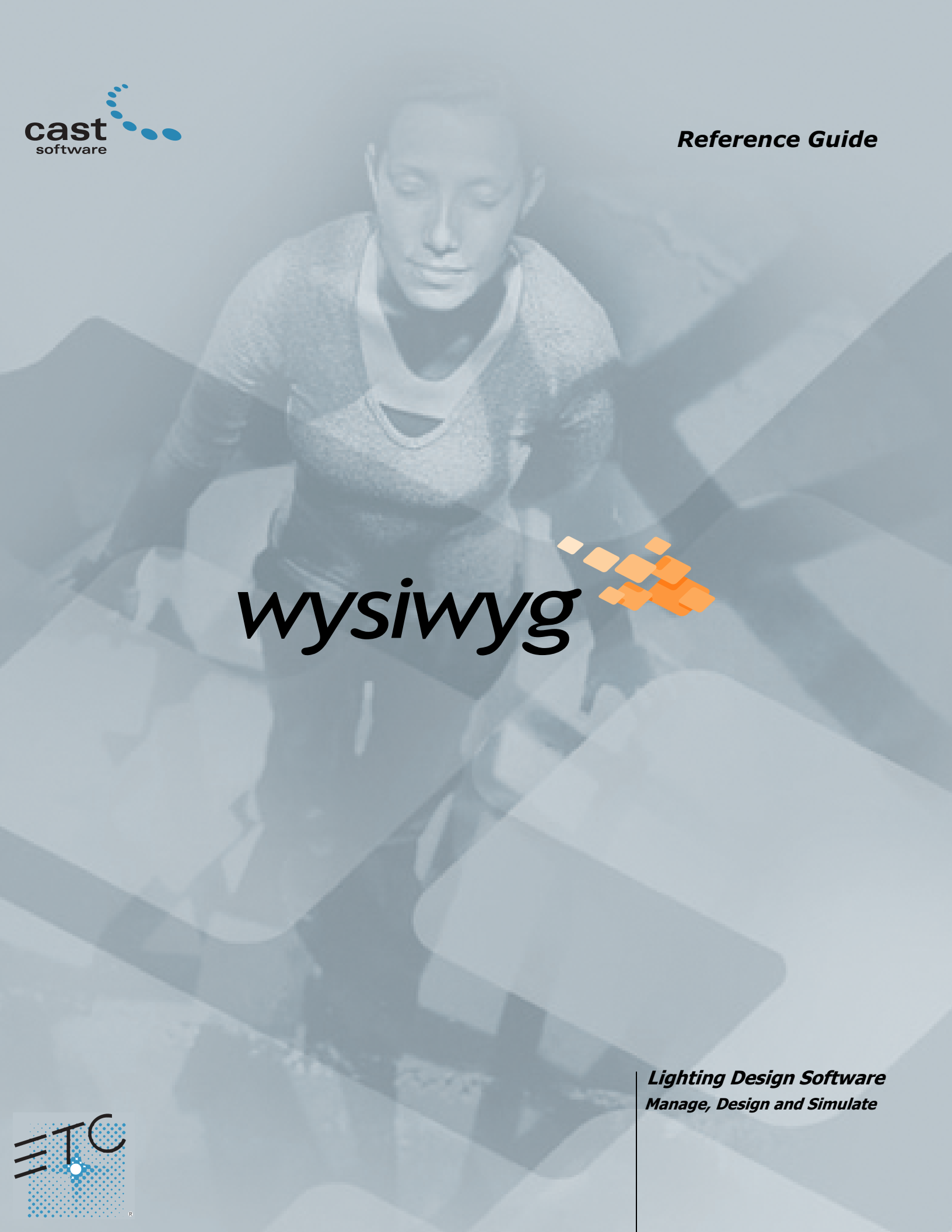
cast software

Reference Guide



wysiwyg

Lighting Design Software

Manage, Design and Simulate

ETC

Reference Guide

Product Release 6.0 September 2003

© Cast Group of Companies Inc., 2002-2003. All rights reserved.
WYSIWYG, Cast Software, WYSIWYG Report, WYSIWYG Perform, WYSIWYG Design and Autofocus are trademarks of the Cast Group of Companies Incorporated. All other trademarks and logos are the property of their respective owners.

Contents

1.0 Introduction 1.0-1

1.1 About This Manual 1.1-1
Introduction 1.1-1
Text conventions 1.1-1
1.2 Troubleshooting 1.2-1
Help from ETC Technical Support 1.2-1
Support for WYSIWYG Perform - Console Edition 1.2-2

2.0 The WYSIWYG Suite of Software Products 2.0-1

2.1 WYSIWYG Report 2.1-1
Software features 2.1-1
Upgrade suggestions 2.1-1
Modes 2.1-1
2.2 WYSIWYG Design 2.2-1
Software features 2.2-1
Upgrade suggestions 2.2-1
Modes 2.2-2
2.3 WYSIWYG Perform 2.3-1
Software features 2.3-1
Upgrade suggestions 2.3-1
Modes 2.3-1
2.4 WYSIWYG Perform - Console Edition 2.4-1
Software features 2.4-1
Upgrade suggestions 2.4-1
Modes 2.4-1
2.5 WYSIWYG Learn 2.5-1
Software features 2.5-1
Upgrade suggestions 2.5-1
Modes 2.5-1
2.6 WYSIWYG Network 2.6-1
Software features 2.6-1
Upgrade suggestions 2.6-1
Modes 2.6-1
2.7 WYSILink 2.7-1
Software features 2.7-1

3.0 Installation, Activation and Registration 3.0-1

3.1 Installation. 3.1-1
Minimum hardware requirements for WYSIWYG Report 3.1-1
Minimum hardware requirements for WYSIWYG Design 3.1-1
Minimum hardware requirements for WYSIWYG Perform . . . 3.1-1
Minimum hardware requirements for WYSIWYG Network /WYSIWYG Learn 3.1-1
Minimum hardware requirements for WYSILink 3.1-2
Installing WYSIWYG 3.1-2
Installing WYSIWYG Learn or WYSIWYG Network. 3.1-3
3.2 Activation and Registration 3.2-1
Activating and registering WYSIWYG software 3.2-1
3.3 Upgrades 3.3-1
3.4 Subscriptions 3.4-1
Obtaining details 3.4-1
Renewing your subscription. 3.4-1
3.5 Passwords 3.5-1

4.0 WYSIWYG User Interface 4.0-1

4.1 Navigation and Screen Set-up 4.1-1
Welcome window 4.1-1
Modes 4.1-1
Layouts 4.1-3
Popup frames 4.1-3
Multiple main frames 4.1-4
4.2 Shortcut Bars 4.2-1
Working with shortcuts 4.2-2
4.3 Toolbars 4.3-1
The menu bar 4.3-1
Toolbars 4.3-1
4.4 Status Bar 4.4-1
4.5 Wireframe Views 4.5-1
Keyboard and mouse control 4.5-1
View shortcuts 4.5-1
Modifying wireframe views 4.5-2
4.6 Shaded Views 4.6-1
Keyboard and mouse control 4.6-1
Modifying shaded views 4.6-1

5.0 Managing Your WYSIWYG Document 5.0-1

5.1 File Menu 5.1-1
New 5.1-1
Open 5.1-1
DWG/DXF Export 5.1-1
Save 5.1-2
Save As 5.1-2
Merge 5.1-3
Close 5.1-4
Print 5.1-4
Print Preview 5.1-6
Print Setup 5.1-6
Send 5.1-6
Recent file shortcuts 5.1-6
Exit 5.1-6
5.2 Options 5.2-1
Application Options 5.2-1
Show Options 5.2-7
User Options 5.2-11
View Options 5.2-16
5.3 WYSIWYG Viewer 5.3-1

6.0 CAD Mode 6.0-1

6.1 Layout Tabs 6.1-1
6.2 The CAD Environment 6.2-1
Scales 6.2-1
Coordinate system and origin 6.2-1
The missing coordinate 6.2-3
Plot types 6.2-4
Ortho 6.2-5
Snaps 6.2-6
Interactive mode 6.2-7
Working with DWG/DXF files 6.2-8
6.3 The Library Browser 6.3-1
Working with the Library Browser 6.3-1
Custom library items 6.3-3
6.4 Drawing Objects 6.4-1
Drawing a venue 6.4-1
Drawing lines 6.4-4
Drawing points 6.4-5
Drawing circles 6.4-5
Drawing arcs 6.4-6

Drawing text labels 6.4-6
Drawing dimension lines 6.4-7
Drawing surfaces 6.4-7
Drawing risers 6.4-7
Drawing cylinders 6.4-8
Drawing spheres 6.4-8
Drawing pipes 6.4-9
Drawing masking 6.4-9
Drawing focus positions 6.4-9
Drawing cameras 6.4-9
Drawing a compass 6.4-10
Saving custom library items 6.4-10
6.5 Editing Objects 6.5-1
Selecting 6.5-1
Grouping/Ungrouping 6.5-1
Moving 6.5-2
Rotating 6.5-3
Mirroring 6.5-3
Object Properties 6.5-3
Textures 6.5-10
Materials 6.5-11
Layers 6.5-13
Scenes 6.5-15
6.6 CAD Tools 6.6-1
Break 6.6-1
Convert 6.6-2
Divide 6.6-4
Extrude 6.6-7
Fillet 6.6-7
Array 6.6-8
Distance 6.6-9
6.7 Hang Structures 6.7-1
Position Manager 6.7-1
Drawing pipes 6.7-3
Pipe properties 6.7-5
Hanging truss 6.7-6
Using Assembly snap with truss 6.7-7
6.8 Hanging and Focusing Fixtures 6.8-1
Hanging fixtures 6.8-1
Custom fixtures 6.8-2
Pipe snap 6.8-2
Focusing fixtures 6.8-4
Drawing focus positions 6.8-4
Focus position properties 6.8-5
Quick focus 6.8-5
Fixture-specific focusing tools 6.8-6
Fixture properties 6.8-9
Fixture notation 6.8-16
Inserting symbols 6.8-19
Symbol properties 6.8-19
6.9 Colour, Gobos, and Accessories 6.9-1
Placing colour and gobos 6.9-1
Colour/gobo lists 6.9-2
Custom gobos 6.9-4
Quick tools 6.9-6
6.10 The Flight Case 6.10-1
6.11 Rendering 6.11-1

7.0 Data Mode **7.0-1**
7.1 Layout Tabs 7.1-1
7.2 Spreadsheets 7.2-1
Data fields 7.2-1
Working in the spreadsheet 7.2-2
Inserting fixtures in Data mode 7.2-4

Inserting focus positions in Data mode 7.2-4
Inserting positions in Data mode 7.2-5
Inserting colour in Data mode 7.2-5
Data filters 7.2-5
Customizing spreadsheets 7.2-6
Creating a new spreadsheet 7.2-7
7.3 Patch 7.3-1
Working in the patch view 7.3-1
Reading the patch 7.3-2
Patch view properties 7.3-3
7.4 Errors 7.4-1
Errors 7.4-1
Error notifications 7.4-2
Error spreadsheet properties 7.4-3

8.0 Design Mode 8.0-1
8.1 Layout Tabs 8.1-1
8.2 The Design Tools 8.2-1
Using the Design tools 8.2-2
8.3 Concepts 8.3-1
Creating concepts 8.3-1
Updating concepts 8.3-1
Working with concepts 8.3-1
8.4 Palettes 8.4-1
Creating palettes 8.4-1
Updating palettes 8.4-1
8.5 Creating Looks 8.5-1
Saving lighting looks 8.5-1
8.6 Rendering 8.6-1
Saving render settings 8.6-1

9.0 Presentation Mode 9.0-1
9.1 Layout Tabs 9.1-1
9.2 Reports 9.2-1
Working with reports 9.2-1
Redesigning headings 9.2-2
Show information 9.2-4
9.3 Plots 9.3-1
Inserting items onto plots 9.3-2
Content and Item properties 9.3-3
Plot items 9.3-4
Image Items 9.3-8
Worksheet, Report, and Spreadsheet Items 9.3-8
Plot Layout Tools 9.3-9
9.4 Images 9.4-1
9.5 Worksheets 9.5-1
9.6 Pipe Tapes 9.6-1

10.0 Live Mode 10.0-1
10.1 Layout Tabs 10.1-1
10.2 Connecting to Consoles 10.2-1
Connecting to a console 10.2-1
Connecting with a Wyg-it 2 . . . . . 10.2-3
Mini Console 10.2-4
10.3 Autofocus 10.3-1
Autofocus tools and menu commands 10.3-1
10.4 AutoPatch 10.4-1
10.5 Simulation Options 10.5-1
10.6 Rendering 10.6-1
10.7 Visualization With WYSIWYG Perform - Console Edition . . 10.7-1

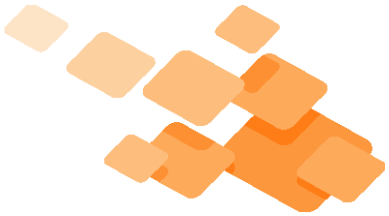
11.0 WYSILink - Link Mode 11.0-1
11.1 Layout Tabs 11.1-1
11.2 Setting up WYSILink 11.2-1
Hardware requirements 11.2-1
Upgrading to WYSILink 11.2-1
ETCNet2 compatibility 11.2-1
11.3 WYSILink Mode 11.3-1
Message Log 11.3-1
Message Log options 11.3-2
Display Log Options 11.3-3
Exporting the Message Log 11.3-5
Printing the Message Log 11.3-5
11.4 ETCLink Functions 11.4-1
The WYSILink toolbar 11.4-1
About dimmer 11.4-1
About dimmer rack 11.4-2
Node Manager 11.4-3
Backup looks 11.4-4
Load recording 11.4-7

A.0 Using the SuperPro Monitoring Tool A.0-1
A.1 SuperPro Monitoring Tool A.1-1
Introduction A.1-1
Changing the protocol A.1-1
Viewing servers on the network A.1-2
Searching for a server A.1-3
Refreshing the server list A.1-3
Additional help A.1-3

Index I-1

Chapter 1.0

Introduction



This chapter introduces you to the WYSIWYG Reference Guide, provides information for troubleshooting, and lists contact information should you need further assistance.

In this chapter

1.1 About This Manual 1.1-1

1.2 Troubleshooting 1.2-1

1.1 About This Manual

Introduction

This manual provides instructions for the installation and operation of the different levels of the WYSIWYG application. WYSIWYG is available at the following levels:

- WYSIWYG Report
- WYSIWYG Design
- WYSIWYG Perform
- WYSIWYG Perform - Console Edition
- WYSIWYG Learn
- WYSIWYG Network

All levels are discussed in this manual. For a detailed description of the levels, refer to Chapter 2.0, "The WYSIWYG Suite of Software Products."

All of the levels use the same user interface, described in Chapter 4.0, "WYSIWYG User Interface." Different tasks are accomplished in various modes. For example, drawing is done in the CAD mode and data editing is done in the Data mode. These modes may have different features depending on the level of WYSIWYG you are running.

Text conventions

The following text conventions are used in this manual:

- Menus and menu commands appear in **Arial bold**. For example, "from the **Edit** menu, choose **Layers**."
- User interface elements such as buttons, tools, shortcuts, and dialog boxes appear in *Tahoma Oblique*. For example, "to draw a riser, click the *Riser* tool on the *Draw* toolbar."
- Keyboard keys are indicated in ALL CAPS. For example, "press the TAB key to enter the missing coordinate."
- References to manuals appear in *italic* font. For example, "for additional information on working with recorded loads, please refer to your *Sensor CEM User Manual*."

1.2 Troubleshooting

If you have problems using your software, please consult the following resources:

- the contents of this manual or the online help available from within WYSIWYG
- the ETC Web site at www.etcconnect.com.
- the Cast Software Web site at www.cast-soft.com

If these resources are insufficient, please contact ETC Technical Support.

Help from ETC Technical Support

For assistance with a problem, contact ETC Technical Support directly at one of the offices identified below.

When calling for help, please have the following information ready:

- Which level you are using (Report, Design, or Perform).
- Software version (from the **Help** menu, choose **About WYSIWYG**)
- What kind of computer you are using (processor and speed, amount of memory, graphics card, operating system.)
- DMX interface type (if applicable).

Americas

ETC International

Technical Support Department

3030 Laura Lane

Middleton, WI 53562

+1-800-688-4116

+1-608-831-4116

wysiwyg@etcconnect.com

Europe

ETC Europe Ltd.

Technical Support Department

5 Victoria Industrial Estate

Victoria Road,

London W3 6UU England

+44 (0)20 8896 1000

wysiwyg@etceurope.com

Asia

ETC Asia, Ltd.

Technical Support Department

Room 605-606

Tower III, Enterprise Square

9 Sheung Yuet Road

Kowloon Bay, Kowloon, Hong Kong

(+852) 2799 1220

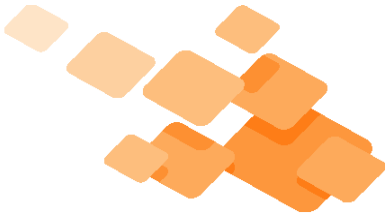
service@etcasia.com

Support for WYSIWYG Perform - Console Edition

For assistance with a problem related to WYSIWYG Perform - Console Edition, contact the manufacturer of the console to which you are connected. The console manufacturer is listed on the Welcome page that appears when you launch the application.

Chapter 2.0

The WYSIWYG Suite of Software Products



This chapter describes the different levels of WYSIWYG.

In this chapter

2.1 WYSIWYG Report	2.1-1
2.2 WYSIWYG Design	2.2-1
2.3 WYSIWYG Perform	2.3-1
2.4 WYSIWYG Perform - Console Edition	2.4-1
2.5 WYSIWYG Learn	2.5-1
2.6 WYSIWYG Network	2.6-1
2.7 WYSILink	2.7-1

2.1 WYSIWYG Report

WYSIWYG Report is the basic WYSIWYG package. It allows lighting professionals to create 2D plans of their productions along with all of the necessary paperwork.

WYSIWYG Report is also available in an educational edition. This product, WYSIWYG Report Educational Edition, is a fully functional version of Report that is available to students and faculty at a special price.

The following specifications are subject to change.

Software features

- 2D plotting of set and lighting designs (plans, sections and elevations)
- Comprehensive 3D library of fixtures, accessories, truss, colors, gobos, props, etc.
- Extensive CAD tools
- Excellent zooming and maneuvering abilities
- Custom layers and scenes for complete control and management of information
- Loading and exporting of DXF, DWG, Excel, BMP, and JPG
- Tabular editing of lighting information including channel, dimmer, color, etc.
- Fully integrated paperwork and 2D CAD systems
- Error checking of production paperwork and hookups
- Reports including instrument counts and schedules, patch sheets, etc.
- Printing of plots and reports up to 36”x48” / A0
- Custom report and plot content and layout
- Ability to print all view types including 2D CAD, reports, spreadsheets, bitmap files, jpeg files, and logos
- Multi-monitor support
- 12 months of software and library updates included

Upgrade suggestions

- A faster processor improves responsiveness.
- Memory requirements depend on the size of the show. Insufficient memory can seriously affect performance. Add memory if there is excessive disk activity.

Modes

The following working modes are available in WYSIWYG Report:

- CAD 2D
- DATA
- PRES 2D

Please refer to the respective chapters of this guide for detailed explanations about the features in each mode.

2.2 WYSIWYG Design

WYSIWYG Design builds on the feature set of Report and adds the ability for lighting professionals to explore their designs in a 3D environment. 3D CAD tools, shaded and isometric views, and photo-realistic rendering tools are added, as well as the ability to create and store images of lighting looks using the integrated design tools.

WYSIWYG Design is also available in an educational edition. This product, WYSIWYG Design Educational Edition, is a fully functional version of Design that is available to students and faculty at a special price.

The following specifications are subject to change.

Software features

- Includes all the features of WYSIWYG Report
- 3D plotting of set and lighting designs
- Multiple working views including isometric
- Comprehensive 3D library of fixtures, accessories, truss, colors, gobos, props, etc.
- Extended library includes textures and materials
- Library objects are multi-layered for realistic texturing
- Objects have variable translucency
- Fully integrated paperwork and CAD systems
- Loading and exporting of DXF, DWG, Excel, BMP, and JPG file types
- Print all view types including 3D views, bitmap files, jpeg files, spreadsheets and logos
- Integrated design mode allows creation of static lighting looks without a console
- Rendering generates a photo-realistic picture of your lighting look
- Rendering wizard provides options for smoke, shadows, ambient light, etc.
- Renderings include bounce and reflection from surfaces
- OpenGL technology for improved beam simulation
- Multi-monitor support
- 12 months of software and library updates included

Upgrade suggestions

- A faster processor improves responsiveness and rendering times.
- Memory requirements depend on the size of the show. Insufficient memory can seriously affect performance. Add memory if there is excessive disk activity.
- A high-performance OpenGL accelerator card significantly improves the performance of real-time simulation.

Modes

The following working modes are available in WYSIWYG Design:

- CAD
- DATA
- DESIGN
- PRES

Please refer to the respective chapters of this guide for detailed explanations about the features in each mode.

2.3 WYSIWYG Perform

WYSIWYG Perform builds on the feature set of Report and Design and adds the ability for lighting professionals to pre-cue entire shows in real time. Users can create their productions in 3D along with all of the necessary paperwork. A console can then be connected to the computer and WYSIWYG Perform will simulate the exact effect of the console's output in real-time. WYSIWYG Perform is available in 500, 1000, 2000 and 5000 channel variants.

The following specifications are subject to change.

Software features

- Includes all the features of WYSIWYG Report and Design
- 3D plotting of set and lighting designs
- Multiple working views including front, back, sides, top, bottom and isometric
- Comprehensive 3D library of fixtures, accessories, truss, colors, gobos, props, etc.
- Fully integrated paperwork and CAD systems
- Live mode for real-time lighting visualization
- Accurate simulation of conventional and automated fixtures
- OpenGL technology for improved beam simulation
- Light emitting surfaces allow simulation of lasers, neon, LEDs, projection screens, etc.
- Easy hook-up to consoles and other DMX sources
- Auto Focus protocol lets WYSIWYG send color and focus data to console
- Integrated design mode allows for creation of lighting looks without a console
- Rendering generates a photo-realistic picture of your cue or lighting look
- Renderings include bounce and reflection from surfaces
- Animated renderings available though certified service providers
- Multi-monitor support
- 12 months of software and library updates included

Upgrade suggestions

- A faster processor improves responsiveness and rendering times.
- Memory requirements depend on the size of the show. Insufficient memory can seriously affect performance. Add memory if there is excessive disk activity.
- A high-performance OpenGL accelerator card significantly improves the performance of real-time simulation.

Modes

The following working modes are available in WYSIWYG Perform:

- CAD
- DATA

- DESIGN
- PRES
- LIVE

Please refer to the respective chapters of this guide for detailed explanations about the features in each mode.

2.4 WYSIWYG Perform - Console Edition

WYSIWYG Perform - Console Edition is a version of WYSIWYG Perform intended for distribution with a specific console or family of consoles. Users purchasing a compatible console will receive this edition of WYSIWYG Perform to create shows and begin pre-cueing activities. The Console Edition of WYSIWYG Perform includes most of the functionality of Perform with certain features restricted or removed. It is available in 500 channel format only.

The following specifications are subject to change.

Software features

- Includes all the features of WYSIWYG Perform other than printing or exporting in any format
- 3D plotting of set and lighting designs
- Multiple working views including front, back, sides, top, bottom and isometric
- Comprehensive 3D library of fixtures, accessories, truss, colors, gobos, props, etc.
- Fully integrated paperwork and CAD systems
- Live mode is restricted to visualizing one patch universe at a time
- Accurate simulation of conventional and automated fixtures
- OpenGL technology for improved beam simulation
- Light emitting surfaces allow simulation of lasers, neon, LEDs, projection screens, etc.
- AutoPatch transmits the entire show patch regardless of channel count
- AutoFocus is limited to the fixtures currently available as part of the 500 channel and patch universe restriction
- Rendering is available in Design and Live modes but is limited to 76,800 pixels

Upgrade suggestions

- A faster processor improves responsiveness and rendering times.
- Memory requirements depend on the size of the show. Insufficient memory can seriously affect performance. Add memory if there is excessive disk activity.
- A high-performance OpenGL accelerator card significantly improves the performance of real-time simulation.

Note: You cannot upgrade from WYSIWYG Report or Design to WYSIWYG Perform - Console Edition. You can however upgrade from WYSIWYG Perform - Console Edition to WYSIWYG Perform at any channel count level.

Modes

The following working modes are available in WYSIWYG Perform - Console Edition:

- CAD
- DATA

- DESIGN
- PRES
- LIVE

Please refer to the respective chapters of this guide for detailed explanations about the features in each mode.

2.5 WYSIWYG Learn

WYSIWYG Learn is a special configuration of WYSIWYG that allows an educational institution to run ten instances of the software on a local area network. The workstations can be configured to run any combination of WYSIWYG Report or WYSIWYG Design. In addition, one workstation can run WYSIWYG Perform 500.

The following specifications are subject to change.

Software features

- Includes all the features of WYSIWYG Report, Design, or Perform
- 2D and 3D plotting of set and lighting designs
- Multiple working views including front, back, sides, top, bottom and isometric
- Comprehensive 3D library of fixtures, accessories, truss, colors, gobos, props, etc.
- Fully integrated paperwork and CAD systems
- Accurate simulation of conventional and automated fixtures
- OpenGL technology for improved beam simulation
- Light emitting surfaces allow simulation of lasers, neon, LEDs, projection screens, etc.
- Easy hook-up to consoles and other DMX sources
- Auto Focus protocol lets WYSIWYG send color and focus data to console
- Rendering generates a photo-realistic picture of your cue or lighting look
- Renderings include bounce and reflection from surfaces
- Animated renderings available though certified service providers
- Multi-monitor support
- 12 months of software and library updates included

Upgrade suggestions

- A faster processor improves responsiveness and rendering times.
- Memory requirements depend on the size of the show. Insufficient memory can seriously affect performance. Add memory if there is excessive disk activity.
- A high-performance OpenGL accelerator card significantly improves the performance of real-time simulation.

Modes

The following working modes are available in WYSIWYG Learn:

- CAD
- DATA
- DESIGN
- PRES
- LIVE

Please refer to the respective chapters of this guide for detailed explanations about the features in each mode.

2.6 WYSIWYG Network

WYSIWYG Network allows you to run as many licenses of WYSIWYG Report, Design, and Perform as you want to over a network. The workstations can be configured to run any combination of these products with the use of a single network dongle.

The following specifications are subject to change.

Software features

- Includes all the features of WYSIWYG Report, Design, and/or Perform
- 2D and 3D plotting of set and lighting designs
- Multiple working views including front, back, sides, top, bottom and isometric
- Comprehensive 3D library of fixtures, accessories, truss, colors, gobos, props, etc.
- Fully integrated paperwork and CAD systems
- Accurate simulation of conventional and automated fixtures
- OpenGL technology for improved beam simulation
- Light emitting surfaces allow simulation of lasers, neon, LEDs, projection screens, etc.
- Easy hook-up to consoles and other DMX sources
- Auto Focus protocol lets WYSIWYG send color and focus data to console
- Rendering generates a photo-realistic picture of your cue or lighting look
- Renderings include bounce and reflection from surfaces
- Animated renderings available though certified service providers
- Multi-monitor support
- 12 months of software and library updates included

Upgrade suggestions

- A faster processor improves responsiveness and rendering times.
- Memory requirements depend on the size of the show. Insufficient memory can seriously affect performance. Add memory if there is excessive disk activity.
- A high-performance OpenGL accelerator card significantly improves the performance of real-time simulation.

Modes

The working modes available in WYSIWYG Network will depend on your product configuration. Please refer to the WYSIWYG Report, Design, or Perform sections accordingly. Detailed explanations of each of the various modes are explained in this guide.

2.7 WYSILink

WYSILink is an add-on for WYSIWYG Report, Design, or Perform and inserts a mode button (Link) to the base product when installed. It acts as an interface to ETCLink and monitors the Sensor line of dimmers. When a problem with a fixture is detected, data about the fixture is logged in Link mode and the image of the affected fixture is highlighted on your plot.

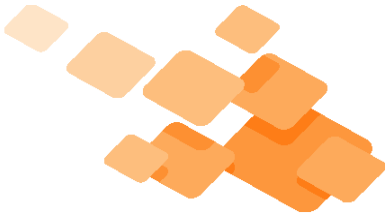
The following specifications are subject to change.

Software features

- Add-on to the features of WYSIWYG Report, Perform, or Design
- System diagnostic fixture data is written to a Message Log
- Ability to record and play backup looks
- Loads can be recorded for specific dimmers

Chapter 3.0

Installation, Activation and Registration



This chapter provides instructions on how to install, activate, and register the WYSIWYG software. You must activate and register the software before you can start using WYSIWYG.

In this chapter

3.1 Installation	3.1-1
3.2 Activation and Registration	3.2-1
3.3 Upgrades	3.3-1
3.4 Subscriptions	3.4-1
3.5 Passwords	3.5-1

3.1 Installation

Follow the procedures in this section to install the level of WYSIWYG that you have purchased.

Minimum hardware requirements for WYSIWYG Report

- Pentium III, or compatible processor, 300MHz or better
- Win98, 2000, ME, XP operating system
- 64MB RAM
- 300MB free disk space
- 800x600 or better resolution
- Win98/ME/XP/2000 - USB or ECP parallel port

Minimum hardware requirements for WYSIWYG Design

- Pentium III, or compatible processor, 550MHz or better
- Win98, 2000, ME, XP operating system
- 128MB RAM
- 300MB free disk space
- 1024x768 or better resolution
- OpenGL - accelerated display adapter with 16MB video memory
- Win98/ME/XP/2000 - USB or ECP parallel port

Minimum hardware requirements for WYSIWYG Perform

- Pentium III, or compatible processor, 800MHz or better
- Win98, 2000, ME, XP operating system
- 256MB RAM
- 300MB free disk space
- 1024x768 or better resolution
- OpenGL - accelerated display adapter with 16MB video memory
- Compatible DMX reception device (except when used with certain consoles - contact the console manufacturer for details)
- MIDI output port to use with Auto Focus
- Win98/ME/XP/2000 - USB or ECP parallel port

Minimum hardware requirements for WYSIWYG Network/ WYSIWYG Learn

- Pentium III, or compatible processor, 300MHz or better
- Win98, 2000, ME, XP operating system
- 64MB RAM
- 2MB free disk space
- Win98/ME/XP/2000 - USB or ECP parallel port
- Network interface card running TCP/IP

For WYSIWYG Perform workstations:

- Use a WYG-it 2 or compatible DMX reception device (except when used with certain consoles - contact the console manufacturer for details)
- MIDI output port to use with Autofocus-enabled consoles

Minimum hardware requirements for WYSILink

- Sensor dimmer rack with Control Electronic Module (CEM), software 3.11 or later.
- ETCNet2 DMX Node with RFU/Link option, software version 2.1.1 or later.

Installing WYSIWYG

The installation procedure for WYSIWYG is the same regardless of the level that you have purchased.

A guide to the installation should appear when the CD is inserted into your PC's CD drive. This guide displays buttons for all the installation options.

- Install WYSIWYG
- Install WYSIWYG Learn (see procedures on page 3.1-3)
- Install WYSIWYG Network (see procedures on page 3.1-3)
- Install Console Edition
- Install Demo Version
- Install Viewer
- Install Tutorials (this option runs the Quick Start Tutorial)
- Support Documentation (this option allows you to open the documentation PDFs).

Click *INSTALL WYSIWYG* to install the software. Follow the steps in the Installation Wizard as prompted.

If the guide does not automatically open:

1. Access the CD-ROM drive of your machine.
2. Double-click the icon for the *Wyginstall* program.
3. Follow the steps in the Installation Wizard.

Installing WYSIWYG Learn or WYSIWYG Network

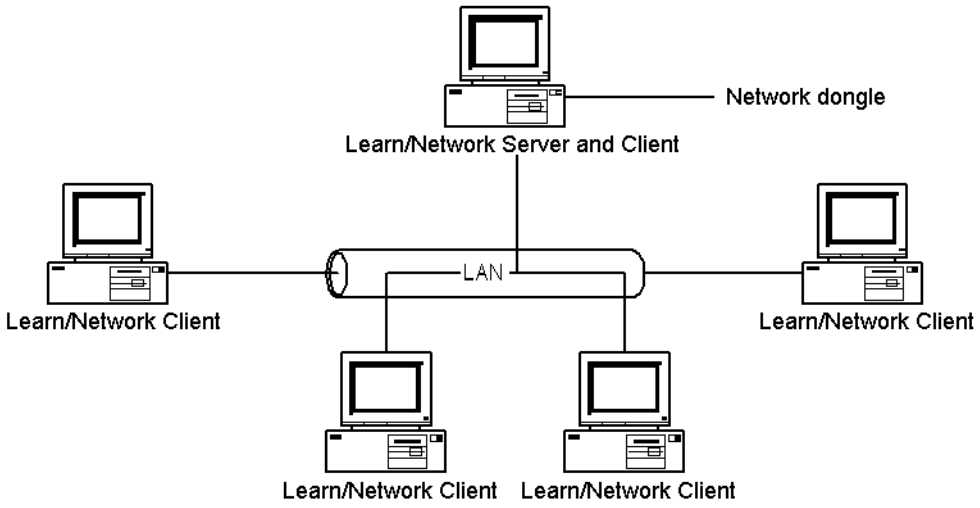
Software components

The WYSIWYG Network and WYSIWYG Learn systems are composed of a server and one or more clients.

The Server is a small application that installs SuperProServer. SuperProServer is a service required to detect the WYSIWYG Learn or Network dongle (a list of current services on a workstation can be viewed under *Control Panel> Administrative Tools> Services*). The Learn or Network Server need only be installed on one computer on the network.

A Learn or Network Client is an add-on for WYSIWYG that enables the software to function over the network. The Client must be installed on all desired workstations. WYSIWYG Learn can run up to 10 clients. The number of clients for WYSIWYG Network is restricted based on the configuration purchased. The standard WYSIWYG installation must be complete before installing the WYSIWYG Learn or Network client(s).

A typical WYSIWYG Network/Learn setup is illustrated in the following diagram. All computers running Network/Learn components must be on the same network (and therefore have the same network ID and subnet mask).



Network services

Windows 2000 and XP

Services are supported in Windows 2000 and XP. Services start running as soon as your computer is turned on, regardless of whether or not the user is logged in. The service will begin automatically each time the workstation is restarted, and will run in the background without any prompting from the user. The service may also be started manually; the choice is available during the installation process.

Windows 98 and ME

Windows 98 and ME do not support services. In this case, the Server will install a small .EXE file for which a shortcut will be created automatically in the start-up folder of the startup menu. When a user logs on, any shortcuts in the start-up folder are triggered instantly. The SuperProServer service will then run in the background until such time as the user logs off. The "service" may be started manually; this choice is available during the installation process.

Network protocol

WYSIWYG Learn or WYSIWYG Network will run on any LAN network that supports TCP/IP protocol (NetBEUI and IPX/SPX networks are not currently supported).

Detecting the network dongle

If the service is not running, the Clients will not be able to detect the dongle and the application will terminate. Ensure that the dongle is attached to the workstation that is running the Server.

Before you begin

- Previous versions of WYSILAB or WYSIWYG Learn/Network should be removed before beginning the install process (if this step is not completed, the install menu will offer the option before continuing with the installation). The workstations will then need to be rebooted for the changes to take effect.
- Before installing the Client software, a regular WYSIWYG application must be installed on each workstation.

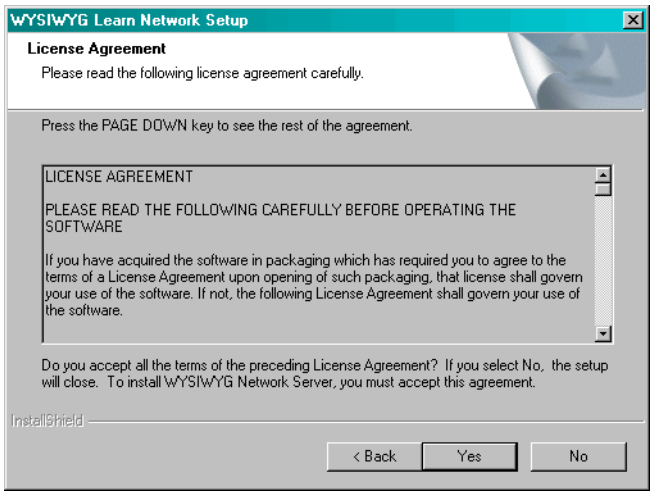
To install the Learn/ Network Server

1 From the *WYSIWYG Install* window, select *INSTALL WYSIWYG LEARN* or *WYSIWYG NETWORK*.

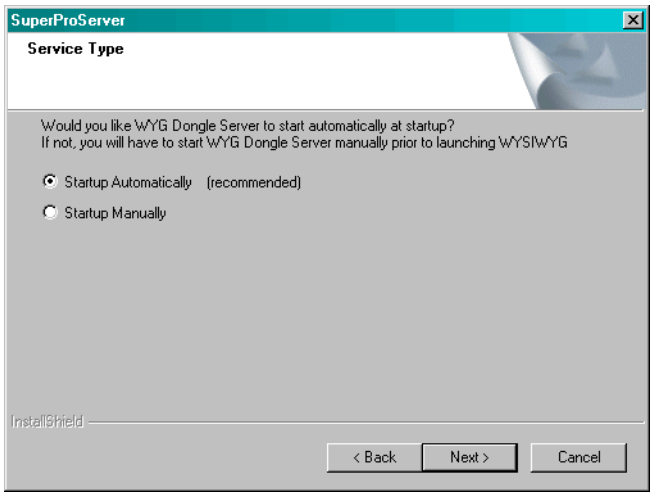
Result: A dialog box will ask you to select which component you wish to install.

2 Select *Install WYSIWYG Network Server* or *Install WYSIWYG Learn Server*.

Note: The Server should only be installed on one computer on the network. This is the computer to which you will attach the network dongle.

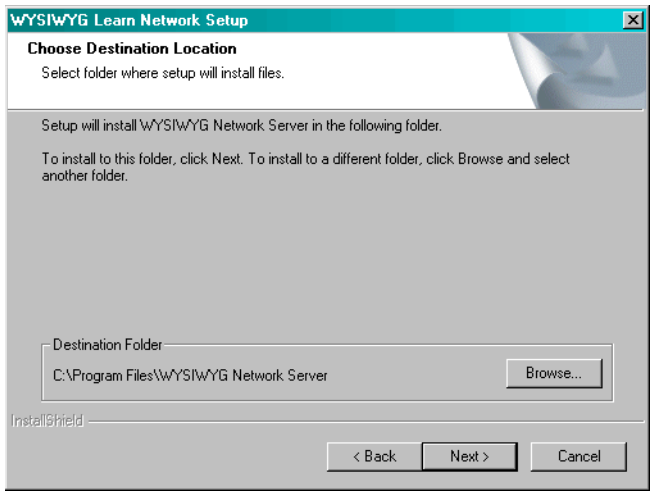

WYSIWYG Learn Network Setup
License Agreement
Please read the following license agreement carefully.
Press the PAGE DOWN key to see the rest of the agreement.
LICENSE AGREEMENT
PLEASE READ THE FOLLOWING CAREFULLY BEFORE OPERATING THE SOFTWARE
If you have acquired the software in packaging which has required you to agree to the terms of a License Agreement upon opening of such packaging, that license shall govern your use of the software. If not, the following License Agreement shall govern your use of the software.
Do you accept all the terms of the preceding License Agreement? If you select No, the setup will close. To install WYSIWYG Network Server, you must accept this agreement.
InstallShield
< Back
Yes
No

3 On the *Setup* window, click *Yes* if you accept the terms of the license.

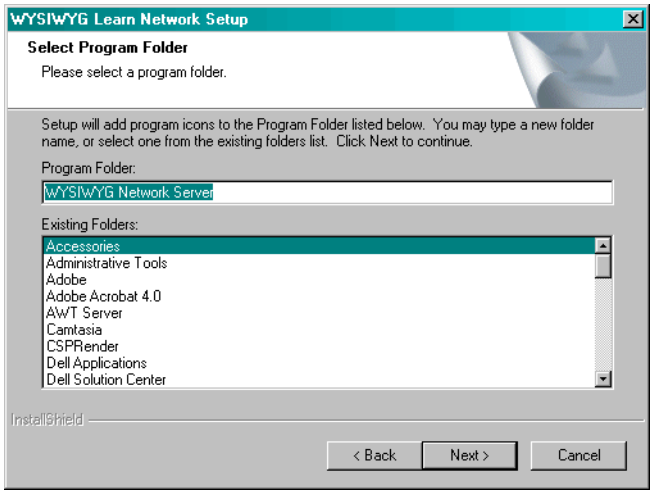

SuperProServer
Service Type
Would you like WYG Dongle Server to start automatically at startup?
If not, you will have to start WYG Dongle Server manually prior to launching WYSIWYG
Startup Automatically (recommended)
Startup Manually
InstallShield
< Back
Next >
Cancel

4 On the *Service Type* window, select the method you want to use to start the SuperProServer service. It is recommended that you select the *Startup Automatically* option.

5 Click *Next*.

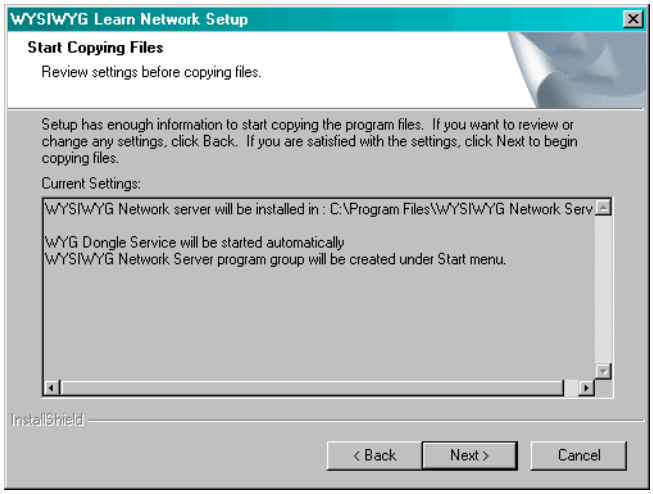

WYSIWYG Learn Network Setup
Choose Destination Location
Select folder where setup will install files.
Setup will install WYSIWYG Network Server in the following folder.
To install to this folder, click Next. To install to a different folder, click Browse and select another folder.
Destination Folder
C:\Program Files\WYSIWYG Network Server
Browse...
InstallShield
< Back
Next >
Cancel

6 On the *Choose Destination Location* window, select a destination location for the Server files, and then click *Next*. The default is C:\Program Files\WYSIWYG Network Server for WYSIWYG Network and C:\Program Files\WYSIWYG Learn for WYSIWYG Learn.



7 On the *Select Program Folder* window, select a program folder for the shortcut. The default is WYSIWYG Network Server or WYSIWYG Learn.

8 Click *Next*.


WYSIWYG Learn Network Setup
Start Copying Files
Review settings before copying files.
Setup has enough information to start copying the program files. If you want to review or change any settings, click Back. If you are satisfied with the settings, click Next to begin copying files.
Current Settings:
WYSIWYG Network server will be installed in : C:\Program Files\WYSIWYG Network Serv
WYG Dongle Service will be started automatically
WYSIWYG Network Server program group will be created under Start menu.
InstallShield
< Back
Next >
Cancel

9 On the *Start Copying Files* window, review the installation settings. Click *Next* to begin the installation.

10 After all the files are copied, click *FINISH* to complete the installation.

To install the Network/ Learn Client

1 From the *WYSIWYG Installation* window, select *INSTALL WYSIWYG LEARN* or *WYSIWYG NETWORK.*

2 Select *Install WYSIWYG Network Client* or *WYSIWYG Learn Client*.

Result: The CAST license screen will be displayed for your review.

Note: WYSIWYG must be installed before you apply this executable.

3 If you accept the terms of the license, click *yes* to continue. The installation will begin.

4 Click *FINISH* to complete the installation.

3.2 Activation and Registration

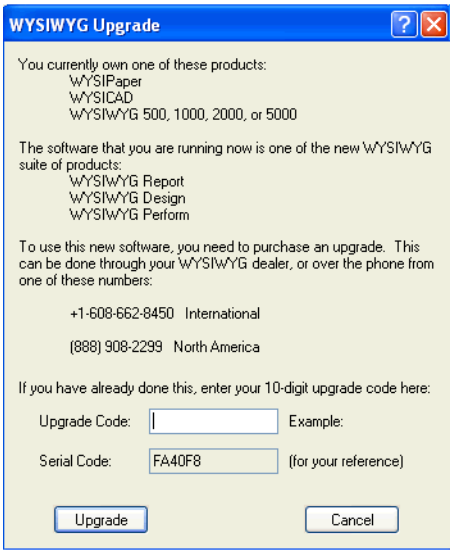
After installing WYSIWYG, you will need to activate and register the software before you can start using it.

Activating and registering WYSIWYG software

Activating the WYSIWYG software involves entering a 10-digit activation code. Registering the software involves entering a validation code.

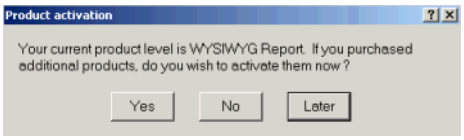
To activate WYSIWYG

1. Run WYSIWYG.
2. If you have purchased a new box product containing a new version dongle skip to step 7.
3. If you previously owned WYSIWYG 3.5 and have upgraded to the new version of the product, the first window that appears is the *WYSIWYG Upgrade* dialog box.


WYSIWYG Upgrade
You currently own one of these products:
WYSIPaper
WYSICAD
WYSIWYG 500, 1000, 2000, or 5000
The software that you are running now is one of the new WYSIWYG suite of products:
WYSIWYG Report
WYSIWYG Design
WYSIWYG Perform
To use this new software, you need to purchase an upgrade. This can be done through your WYSIWYG dealer, or over the phone from one of these numbers:
+1-608-662-8450 International
(888) 908-2299 North America
If you have already done this, enter your 10-digit upgrade code here:
Upgrade Code:
Example:
Serial Code:
FA40F8
(for your reference)
Upgrade
Cancel

The code you enter in this dialog will enable your old dongle to function with the new version of WYSIWYG.

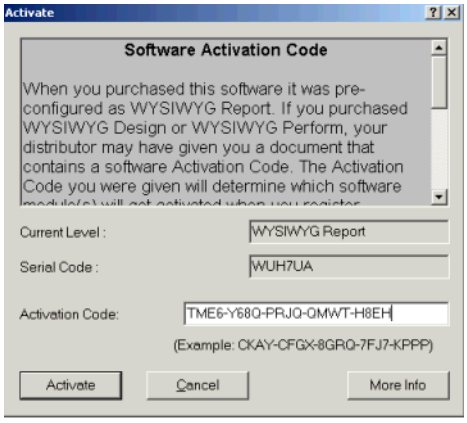
4. Enter your upgrade code and click *Upgrade*.
5. If you upgraded to WYSIWYG Report continue to step 7.
6. If you upgraded to a higher level product - Design or Perform - skip to "To register WYSIWYG" on page 3.2-2.
7. If you have purchased a new box product containing a new version dongle or have upgraded from 3.5 to WYSIWYG Report, the *Product activation* dialog box is displayed.


Product activation
Your current product level is WYSIWYG Report. If you purchased additional products, do you wish to activate them now ?
Yes
No
Later

By default, your dongle may be set to run WYSIWYG Report. If you purchased WYSIWYG Report, click *No* and proceed to "To register WYSIWYG" on page 3.2-2.

If you purchased a higher level product - Design or Perform, click *Yes* and proceed to step 8.

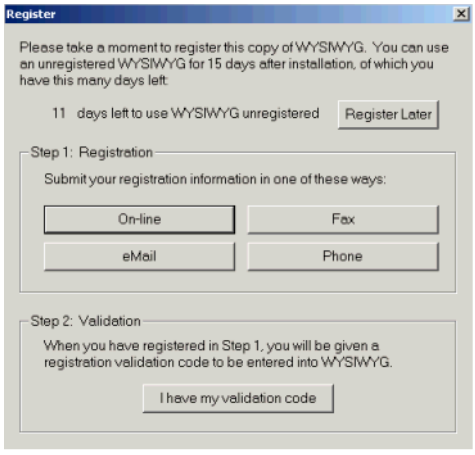
8 If you selected *Yes* in step 7, the Activate dialog box is displayed.


Activate
Software Activation Code
When you purchased this software it was pre-configured as WYSIWYG Report. If you purchased WYSIWYG Design or WYSIWYG Perform, your distributor may have given you a document that contains a software Activation Code. The Activation Code you were given will determine which software
Current Level :
WYSIWYG Report
Serial Code :
WUH7UA
Activation Code:
TME6-Y68Q-PRJQ-QMWT-H8EH
(Example: CKAY-CFGX-8GRQ-7FJ7-KPPP)
Activate
Cancel
More Info

9 To activate your dongle to run the higher level product that you purchased - Design or Perform - enter the Activation code given to you at the time of purchase and click *Activate*. Proceed to "To register WYSIWYG" on page 3.2-2.

To register WYSIWYG

1 Once your product is successfully activated, you will be prompted to register. The *Register* dialog box is displayed.


Register
Please take a moment to register this copy of WYSIWYG. You can use an unregistered WYSIWYG for 15 days after installation, of which you have this many days left:
11 days left to use WYSIWYG unregistered
Register Later
Step 1: Registration
Submit your registration information in one of these ways:
On-line
Fax
eMail
Phone
Step 2: Validation
When you have registered in Step 1, you will be given a registration validation code to be entered into WYSIWYG.
I have my validation code

You may postpone registration for a period of up to 15 days after installation.

To register later, click *Register Later*. You must register and obtain a Registration Validation Code in order to continue using the product after this time. It is recommended to register as soon as possible to avoid any restriction from using the product.

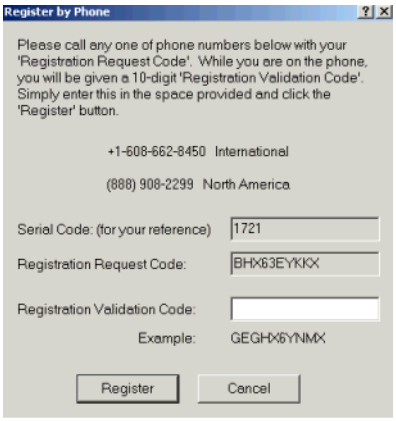
You can register by phone, online, fax, or email.

2 To register by phone, click *Phone* and proceed to step 3.

To register by any of the other methods, select the appropriate button and follow the instructions as prompted.

When you have obtained your validation code, click *I have my validation code* and proceed to step 4.

3 If you have chosen to register by phone, the *Register by Phone* dialog box is displayed.


Register by Phone
Please call any one of phone numbers below with your 'Registration Request Code'. While you are on the phone, you will be given a 10-digit 'Registration Validation Code'. Simply enter this in the space provided and click the 'Register' button.
+1-608-662-8450 International
(888) 908-2299 North America
Serial Code: (for your reference)
1721
Registration Request Code:
BHX63EYKKX
Registration Validation Code:
Example:
GEGHX6YNMX
Register
Cancel

Follow the instructions in the dialog, enter your validation code and click *Register*. A successful registration completes the registration and activation process.

4 If you selected *I have my validation code* in step 2, the *Insert Registration Code* dialog box is displayed.

5 Enter your validation code and click *OK*. A successful registration completes the registration and activation process.

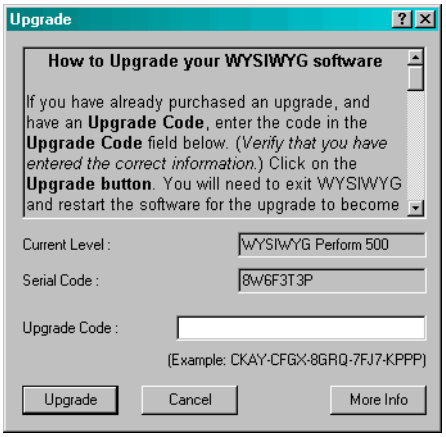
3.3 Upgrades

Increasing the level of your software is as easy as purchasing an upgrade. An upgrade code will enable you to run the advanced features of a higher level product - Design or Perform - using the same dongle you already own. Contact your local ETC WYSIWYG dealer and purchase an upgrade. You will be given an upgrade code at the time of purchase.

To upgrade the level of your WYSIWYG product

1 Run WYSIWYG.

2 From the **Help** menu, choose **Upgrade**.


Upgrade
How to Upgrade your WYSIWYG software
If you have already purchased an upgrade, and have an Upgrade Code, enter the code in the Upgrade Code field below. (Verify that you have entered the correct information.) Click on the Upgrade button. You will need to exit WYSIWYG and restart the software for the upgrade to become
Current Level :
WYSIWYG Perform 500
Serial Code :
8W6F3T3P
Upgrade Code :
(Example: CKAY-CFGX-8GRQ-7FJ7-KPPP)
Upgrade
Cancel
More Info

The current level of WYSIWYG installed on your machine is displayed in the *Current Level* box. The serial code of your dongle is shown in the *Serial Code* box.

3 In the *Upgrade Code* box, enter your upgrade code and then click *Upgrade*.

4 Exit WYSIWYG and then restart your computer.

5 Launch WYSIWYG.

Result: The new level of software is displayed in the upper right corner of the WYSIWYG user interface.

Note: If you would like to purchase an upgrade but are unsure of who to contact, click *More Info* to obtain a list of ETC dealers.

3.4 Subscriptions

Your subscription to WYSIWYG is valid for a specific period of time. You can easily view and renew your subscription from within WYSIWYG.

Obtaining details

Details of your subscription are stored within WYSIWYG.

To obtain details

1 From the **Help** menu, choose **About WYSIWYG**.

Result: The month and year that your subscription expires is displayed in the *Subscription Expire Date* box.

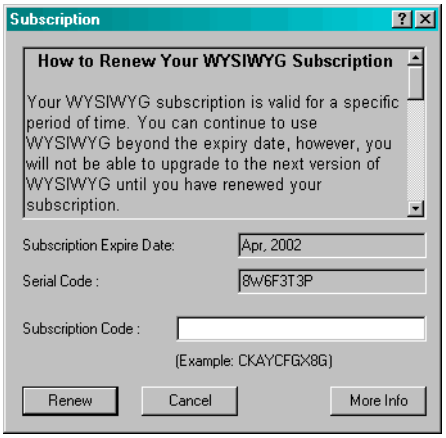
Renewing your subscription

Your WYSIWYG subscription is valid for a specific period of time. You can continue to use WYSIWYG beyond your subscription expiry date, however, you will not be able to access the quarterly software updates until you renew your subscription.

Contact your local ETC dealer to purchase a renewal for your subscription. ETC will provide you with the subscription code that you require.

To renew

1 From the **Help** menu, choose **Renew Subscription.**


Subscription
How to Renew Your WYSIWYG Subscription
Your WYSIWYG subscription is valid for a specific period of time. You can continue to use WYSIWYG beyond the expiry date, however, you will not be able to upgrade to the next version of WYSIWYG until you have renewed your subscription.
Subscription Expire Date:
Apr, 2002
Serial Code :
8W6F3T3P
Subscription Code :
(Example: CKAYCFGX8G)
Renew
Cancel
More Info

The date when your WYSIWYG subscription expires is displayed in the *Subscription Expire Date*.

2 In the *Subscription Code* box, enter the code. The code is 10-digits and is alphanumeric.

3 Verify that you have entered the correct information.

4 Click *Renew*.

Exit WYSIWYG and restart the software for the renewal to become effective.

Note: If you would like to renew your subscription but are unsure of who to contact, click *More Info* to obtain a list of ETC dealers.

3.5 Passwords

You can set a password in WYSIWYG to password protect your dongle.

To set a password

1 From the **Help** menu, choose **Password**.

Result: The *Change Password* dialog box is displayed.

2 Enter your new password in the *New password* box. The password can be any combination of letters, numbers, and characters. As you type the password, WYSIWYG displays an asterisk (*) for each character you type. The length of the password is between 6 and 20 characters.

3 Re-enter the password in the *Confirm password* box.

4 Click *OK*.

Result: The next time that you launch WYSIWYG on the same machine, you will be prompted to enter your password. Enter your password in the *Enter password* box. If you do not want to be prompted to enter a password every time you launch WYSIWYG, select the *Remember for next time* check box.

To change a password

1 From the **Help** menu, choose **Password**.

Result: The *Change Password* dialog box is displayed.

2 Enter the password you have previously set in the *Current password* box. You must enter the current password if you want to change the password. As you type the password, WYSIWYG displays an asterisk (*) for each character you type.

3 Enter a new password in the *New password* box.

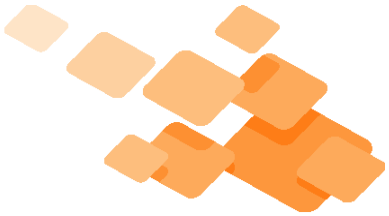
4 Re-enter the password in the *Confirm password* box.

5 Click *OK*.

Note: If you decide you do not want to password protect your dongle, select the *No password* check box.

Chapter 4.0

WYSIWYG User Interface



All of the WYSIWYG levels use the same set of screens. The user interface was designed to facilitate smooth transitions from mode to mode, and to keep the data organized and easy to find.

In this chapter

4.1 Navigation and Screen Set-up	4.1-1
4.2 Shortcut Bars	4.2-1
4.3 Toolbars	4.3-1
4.4 Status Bar	4.4-1
4.5 Wireframe Views	4.5-1
4.6 Shaded Views	4.6-1

4.1 Navigation and Screen Set-up

Welcome window

When you launch WYSIWYG, the *Welcome* window is displayed and the application level is shown in the upper right corner of the window.

From the WYSIWYG menu, you can choose to open an existing file or create a new file. A list of template files are shown on the left and a list of recently saved files are shown on the right.

Click a file name to start working with WYSIWYG.

Modes

The mode buttons located along the top of the user interface are used to access the different working modes available within the WYSIWYG levels.

Click the appropriate mode button to change modes.

CAD
CAD mode is where you create your show drawings. This includes drawing your venue, set pieces, lighting positions, focus positions and lighting fixtures. CAD mode operates like many other CAD programs, so many of the concepts will be familiar to those who have used a computer-aided drafting program before. WYSIWYG adds features that are specific to the entertainment industry, such as a comprehensive 3D library containing truss, lighting equipment and accessories, as well as props, musical instruments and various human figures.

In WYSIWYG Report, you are limited to 2D views of your plot. In Design and Perform, 3D isometric and shaded views are available.

Data
As you draw in CAD mode, all of your show information is compiled into spreadsheets which are accessed in Data mode. Here you can view and edit the data for all your fixtures. Data mode operates like many other spreadsheet programs, so many of the concepts will be familiar to those who have used a spreadsheet program. The CAD and Data modes remain completely interactive and any information modified in either mode will be updated in the other.

Design
Design mode was created to provide lighting designers with an avenue for experimentation. Design mode can be used as a troubleshooting tool or as a creative tool to help you come up with cue concepts. In Design mode, you can create static lighting looks using the design tools, save and render those looks to output photo-realistic pictures. Fixtures can be turned on and controlled without having to patch or connect to a console.

Presentation
The Presentation (Pres) mode contains all of the tools necessary for creating professional printouts of your show document including reports, plots, and images. As with all of the other modes, the contents of the Pres mode are continually updated as you draw, input data, and modify your show file. In addition to being able to create your own, WYSWIYG provides a series of default plots and reports that are ready for printout. These defaults can be used as is or customized to suit your preferences.

Live
Live mode is used for graphically simulating the output of a lighting control console or compatible offline editor. This is where you can pre-cue and visualize your show. Live mode also has rendering ability so that you can create photo-realistic pictures of the simulated lighting looks. Unlike Design mode, Live mode will display transitions from cue to cue, allowing you to see the programmed movement of light over time.

Link
The WYSILink (Link) mode becomes available when you purchase and install the WYSILink add-on for WYSIWYG. WYSILink acts as an interface to ETCLink and monitors the Sensor line of dimmers. When a problem with a fixture is detected, data about the fixture is logged in Link mode and the image of the affected fixture is highlighted on your plot. Link mode is also where you can record and play backup looks, and view information on dimmers and dimmer racks in your system.

Availability of modes
The following working modes are available in WYSIWYG Report:

- CAD 2D
- DATA

- PRES 2D

The following working modes are available in WYSIWYG Design:

- CAD
- DATA
- DESIGN
- PRES

The following working modes are available in WYSIWYG Perform:

- CAD
- DATA
- DESIGN
- PRES
- LIVE

Note: Emphasis adds the mode button Emphasis. WYSILink adds the mode button Link.

Layouts

Beneath the work area in each mode is a series of layout tabs. A layout is a configuration of work views. Views and therefore layouts are mode specific. The layouts are not editable and there are no options for saving custom layouts. To this end, WYSIWYG supports pop-out frames and has multi monitor functionality thus providing alternative tools for customizing your work environment. These features are discussed below.

Click the appropriate layout tab to change layouts.

Views

Views are the windows in the work space. The windows are pre-configured on the screen depending on the layout as discussed above.

The WYSIWYG views include: wireframe, wireframe-selection-only, flight case, shaded, spreadsheet, patch, error, report, image, plot, and worksheet.

The layouts and views available in each mode are discussed at the beginning of each chapter.

Wireframe views are discussed on page 4.5-1 of this chapter. Shaded views are discussed on page 4.6-1.

Popup frames

It is possible to place a view in a "popup frame". Popup frames remain active across modes. For example, if the patch view is placed in a popup frame, it will remain visible even if the mode is changed from Data to CAD.

To place a view in a popup frame

1 Make the desired window active.

2 From the **Options** menu, choose **Open in Popup Frame**.

Tip: Alternatively, click the *Frame Popup* tool on the *View* toolbar.

Result: A new window containing a copy of the selected view will open.

Popup frames retain all the functionality of the original view so you can work in the popup view if you want to. Popup frames are fully integrated with the remaining show document and are continually updated and vice versa. Popup frames can be resized and moved to any location on the screen or across monitors.

To maximize a popup frame

1. Make the popup window active.
2. Click the *Maximize* button along the top of the window.

   Result: The popup window is expanded to fill the working area of the screen, leaving the title bar visible.

To display popup frames in full screen mode

1. Make the popup window active.
2. Click the *Full Screen* button.

   Result: The popup is expanded to fill the entire screen with no title bar visible.
3. To return to popup frame mode, press ESC.

To rollup a popup frame

1. Double-click on the title bar of the popup window.

   Tip: You can also click the *Minimize* button.

   Result: The title bar only is displayed.

Multiple main frames

A main frame is defined as the user interface for a given file. WYSIWYG allows multiple occurrences of the main frame to be opened for a given file. At no time may more than one file be open. Multiple main frames allow the user to work on the same file in multiple modes at the same time.

The additional main frames can be closed using the *Close* tool at the top right corner of the screen. Only the initial main frame will trigger a file closure when this button is pressed.

For increased flexibility, WYSIWYG supports multiple monitors. Main frames and popup frames can be carried across monitors.

To open a new main frame

1. At any time, from the **Options** menu, choose **New Window**.
2. Drag the title bar of the new window to place it where you want it. To make the window bigger or smaller, use the resize tab in the lower right corner of the window or the minimize and maximize tools in the top right.
3. Repeat to open more new windows.

Each window or main frame functions independently but all of the show drawings and data are fully integrated.

4.2 Shortcut Bars

Shortcuts are created for quick access to library items and view types among other things. The shortcut bars are located on the left side of your screen. Shortcut bars are mode and layout specific. Each layout has designated shortcut bars.

In general, there are three types of shortcuts:

- navigation
- library
- views/tools

The first type of shortcut is used for navigation purposes. The navigation shortcuts are:

- Navigation
- Views

Navigation shortcuts

Navigation shortcuts store the mode and layout destination. For example, you can save a navigation shortcut to Data mode / Patch layout. The next time you want to go back to that location, click the shortcut to save you at least one step.

View shortcuts store the plot type and zoom level. View shortcuts only apply to wireframe views.

Library shortcuts

The second type of shortcut is used in place of browsing the library. The library shortcuts are:

- Fixtures
- Accessories
- Colour
- Gobo
- Library
- Truss

Views/tools shortcuts

The final type of shortcut is used to open and save views and tools. The views/tools shortcuts are:

- Columns
- Patch
- Errors
- Worksheets
- Images
- Plots
- Reports
- Looks
- Concepts
- Palettes
- Render

Working with shortcuts

Generally the procedure for creating, modifying, and working with shortcuts is the same regardless of the type of shortcut.

To create a new shortcut

1. Right click in open space on the appropriate shortcut bar and choose the appropriate **New option**.

   Note: For navigation type shortcuts, ensure that the destination is setup and then proceed with step 1.
2. Depending on the type of shortcut you will be prompted to select the object or type a name for the new shortcut.
3. Click *Insert* or *OK*.

   Result: The shortcut will be created and added to the bottom of the list on the shortcut bar selected in step 1.

To remove a shortcut

1. Right click on the shortcut icon that you want to remove.
2. Choose **Delete**.

   Result: You are prompted to confirm the deletion.
3. Click *OK* to confirm.

   Result: The selected shortcut is deleted.

To rename a shortcut

1. Right click on the shortcut icon that you want to rename.
2. Choose **Rename**.
3. Enter a new name for the shortcut.

4 Click OK.

Result: The selected shortcut is renamed.

To clone a shortcut

1 Right click on the shortcut icon that you want to clone.

2 Choose **Clone Shortcut**.

3 Enter a name for the copy.

Result: The selected shortcut is cloned and the copy is added to the bottom of the list.

To arrange shortcuts

You can arrange shortcuts in the shortcut bar area by using drag-and-drop functionality.

1 Click on the shortcut you want to move.

2 While holding down the mouse button, press the ALT key to move the shortcut to the desired location.

3 Release the mouse button to place the shortcut.

To set the display size for shortcuts

1 Right-click in open space on the appropriate shortcut bar and choose either **Icons** or **List** (depending on your current setup).

Result: Choosing **Icons** will display all shortcuts as icons. **List** will cause the shortcuts to be listed in the shortcut bar area, thereby allowing you to fit more shortcuts in the area.

The settings are retained for all shortcuts.

Note: To scroll to the bottom of a long list of shortcuts, use either the scroll button on your mouse or the scroll bar along the side of the shortcut bar.

To use a shortcut

Click on the shortcut icon.

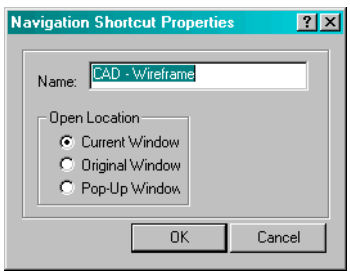
As with all objects in WYSIWYG, all shortcuts have properties that affect the object being inserted, the view or tool, or the navigation method.

To modify a shortcut's properties

1 Right click on the shortcut icon that you want to modify.

2 Choose **Properties**.

Result: The properties dialog box is displayed. The settings modified here will affect the object, view, tool, or navigation destination of the selected shortcut. An example of the *Navigation Shortcut Properties* dialog box is shown below.



3 Enter the new name of the shortcut in the *Name* box.

4 To open the shortcut in the current active window, enable the *Current Window* radio button.

5 To open the shortcut in the main frame in which the shortcut was created, enable the *Original Window* radio button. This option is only applicable if you have multiple main frames (multiple monitors) open.

6 To open the shortcut in a pop-up frame, enable the *Pop-Up Window* radio button. For more information on popup frames, refer to "Popup frames" on page 4.1-3.

4.3 Toolbars

The menu bar

The menu bar is located within the WYSIWYG user interface, below the mode buttons and above the work area. The available menus will change as needed when moving from mode to mode. The individual menus will be explained within the context of each mode chapter.

The menu bar is dockable. It has a grab bar on the left that is used to move the menu bar around the screen. It may be placed on the edges of the work area; top, bottom, left or right. It may also be dragged off the edge of the work area and into its own window. This window remains on top of the WYSIWYG screen and may be dragged anywhere on your display.

Menu commands can be accessed using a mouse, keyboard, or by using hot keys.

Hot keys are explained in “Hot Keys” on page 5.2-15.

To access menu commands using the keyboard

1 Press ALT+*n,* where *n* is the underlined letter in the menu name.

Result: The menu will be displayed.

2 Press the key corresponding to the underlined letter in the command that you want to execute.

Some commands have shortcuts that do not require menu selection. In those cases, the shortcut keys are listed to the right of the command in the menu.

Example:
To undo the last command, press CTRL+Z.

Toolbars

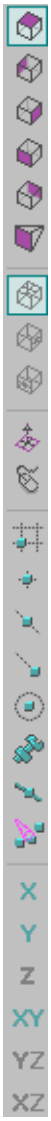
Toolbars provide button access to most commands. This is in lieu of selecting the commands through the menus in the menu bar.

Toolbars, like menus, are mode sensitive. However, unlike menus, you can customize how the toolbars are displayed and which toolbars are open for each mode. Toolbars that are greyed out in the toolbar selection list are not used in the current mode.

Toolbars are typically found directly under the menu bar and have a grab bar on the left that is used to move the toolbar around the screen. They can be placed on the edges of the work area (top, bottom, left or right), and can also be dragged off the edge of the work area and into their own window. This window remains on top of the WYSIWYG screen and can be dragged anywhere on your display.

The following toolbars are available (listed in alphabetical order).

CAD Options toolbar



Plan
Left
Right
Front
Back
Isometric
WorkPlane Plan
WorkPlane Side
WorkPlane Front
Missing coordinate
Interactive
Grid Snap
Points
MidPoints
EndPoints
Centre
Assembly
Pipe
Focus
OrthoX
OrthoY
OrthoZ
OrthoXY
OrthoYZ
OrthoXZ

ComEdit toolbar

Edit box

Data toolbar

Fixture

Colour Select

Gobos

Accessory

Find

Replace

Quick Tools

Design toolbar

Intensity and Focus

Intensity Designer Tool

Focus Designer Tool

Iris Designer Tool

Zoom Designer Tool

Colour Designer Tool

Gobo Designer Tool

Prism Designer Tool

Render Wizard

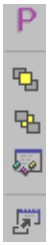
Draw toolbar

Point
Solid Line
Centered Line
Hidden Line
Dotted Line
Spline
Circle
Arc
Text Label
Dimension
Surface
Extruded Surface
Riser
Cylinder
Sphere
Library
Pipe
Truss
Fixture
Accessory
Focus Position

Edit toolbar

Item Properties
Cut
Copy
Paste
Delete
Undo
Redo
Repeat last command

Edit (Pres) toolbar



Item Properties

Bring to Front

Bring to Back

Content Properties

Open Contents

Insert (Pres) toolbar

Line

Rectangle

Round Rectangle

Ellipse

Polygon

Text

Bitmap

Plot item

Spreadsheet item

Report item

Worksheet item

Layers toolbar

Layers

Current Layer

Live toolbar

Intensity and focus

Intensity Designer Tool

Focus Designer Tool

Iris Designer Tool

Colour Designer Tool

Message Log Options toolbar

Message Log Options

Display Log Options

Export

Print

Print Preview

Scenes toolbar

Scenes

Current Scene

Standard toolbar

Save

New document

Open

Print Preview

Print

View Options

Help

Tools toolbar

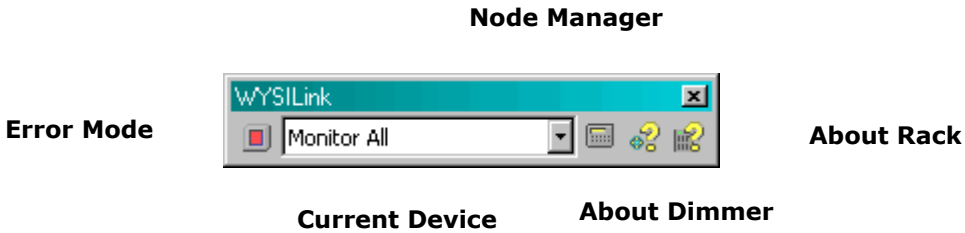
Group
Ungroup
Move
Rotate
Mirror
Break
Convert
Divide
Extrude
Fillet
Distance
Origin
Quick Tools
Quick Focus
Dimension

Universes toolbar

Universe

View toolbar

WYSILink toolbar



To hide and display toolbars using the toolbar list

1. Right-click in the toolbar area.

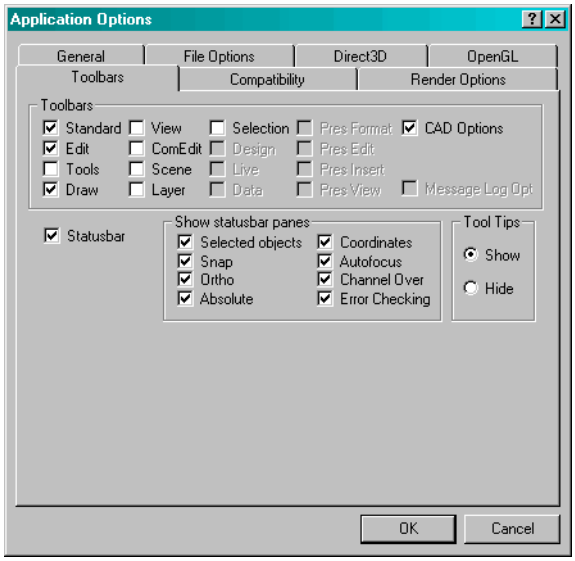
   Result: The toolbar list is displayed.

2. Click on the name of the toolbar you wish to display. A check mark to the left of the toolbar name indicates it is currently displayed.
3. Repeat to display or hide multiple toolbars.

   Tip: From the toolbar list, select *Toolbars* to open the *Application Options* dialog box.

To hide and display toolbars using the Options menu

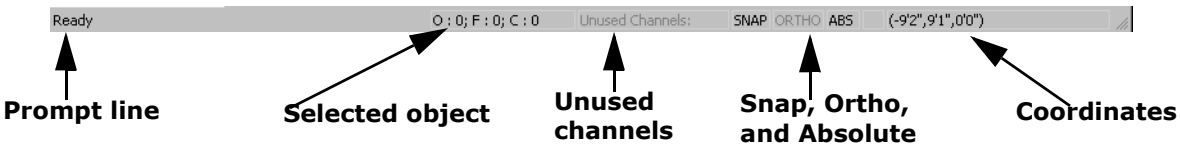
1. From the **Options** menu, choose **Application Options**.
2. Click the *Toolbars* tab.



3 Select the check boxes of the toolbars that you want to display. Toolbars are mode-sensitive. For more information on the *Toolbar* tab, refer to "Toolbars" on page 5.2-4.

4.4 Status Bar

The status bar is displayed along the bottom of the WYSIWYG screen, below the layout tabs. The status bar displays the prompt line, the number of unused channels (WYSIWYG Perform), selected object information, and tracks the status of the snap, ortho and absolute coordinates commands. It also tracks and displays the coordinates of the cursor.



The status bar can be turned on or off from the toolbar list. You can also customize the contents of the status bar from the Application Options window. For more information on setting the options of your drawing, refer to “Application Options” on page 5.2-1.

Prompt line

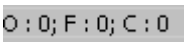
The prompt line displays the current status of a command within your drawing. If you are currently working with a command that requires multiple steps (the placement of a pipe, for example), the prompt line displays a message indicating the next step required to accomplish that task. The prompt line will also display a short description of a command when you point to it using your mouse.

Unused channels

Although there is never a limitation on how many fixtures can be patched in a file, there is a limitation on the number of DMX channels that can be simulated in the Live mode. WYSIWYG will not simulate the DMX values being received on channels exceeding the Perform channel count of your system. WYSIWYG Perform is available in 500, 1000, and 2000+ channels. The unused channels that display on the status bar tracks the remaining channels for simulation based on the level of your Perform system.

Selected object

In this display, “O” stands for the number of objects currently selected. The “F” stands for the number of fixtures selected and the “C” stands for the number of circuits currently selected.



Snap, Ortho and Absolute

The snap, ortho, and absolute entry shows the status of these items. If the indicator is black, a snap or ortho setting is active. If the indicator is gray, snap and ortho are not active.

Absolute coordinates is the default setting, however, you can change this to the Relative mode. This setting determines how coordinate information is displayed mid-command. In ABS, which is the default mode, the coordinates display will always show the coordinates currently under your cursor. In REL, the coordinates display will show the distance and direction you have moved since the last point.

Coordinates

Coordinates are displayed in the order of X, Y and Z. The coordinates display the position of the cursor as a distance from the origin of the drawing. For more information on coordinates and setting the origin point, refer to “Coordinate system and origin” on page 6.2-1.

4.5 Wireframe Views

When you first create a new show document and enter CAD mode, WYSIWYG will default to the Wireframe view. All drawing is done in Wireframe views. Use the Quad view to display three Wireframe views and a shaded view of your drawing.

Keyboard and mouse control

To modify the point of view

- The arrow keys will move you in their respective directions.
- The PAGE UP and PAGE DOWN keys zoom you in and out.
- In a 3D perspective view (isometric or shaded), the CTRL key in combination with any of the aformentioned keys will rotate you around your drawing.
- At any time you can use the SHIFT key in combination with any of the movement keys to move in smaller increments.
- In shaded views, the mouse will substitute all key strokes. Click and drag to pan around and use the mouse wheel for zooming.

View shortcuts

View shortcuts record the plot type as well as the zoom and pan settings of a specific point of view. You can use the shortcut to get back to that exact view point the next time you need to.

To record a view shortcut

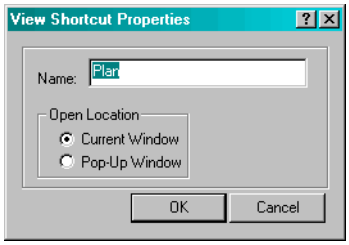
1. Set up the active window for the plot type, zoom, and pan settings you wish to record.
2. Right click on the *Views* shortcut bar and choose **New View**.
3. Enter a name for the new shortcut and click *OK*.

   Result: The shortcut is recorded and available on the *Views* shortcut bar.

To modify the properties of a view shortcut

1 Right click on the Views shortcut icon you want to modify and select **Properties**.

Result: The *View Shortcut Properties* dialog box is displayed.



2 Enter the new name of the shortcut in the *Name* box.

3 To open the shortcut in the current active window, enable the *Current Window* radio button.

4 To open the shortcut in a pop-up frame, enable the *Pop-Up Window* radio button. For more information on popup frames, refer to "Popup frames" on page 4.1-3.

Modifying wireframe views

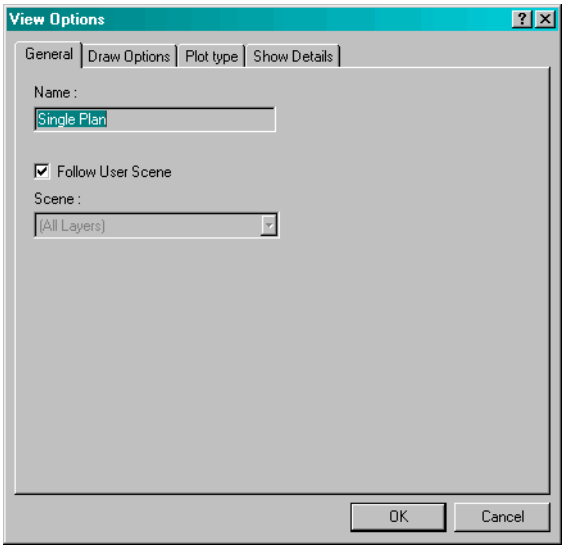
The properties of a wireframe view affect how objects are drawn and how much information is visible. These settings are modified in the View Options.

To modify a wireframe view

1 Ensure the wireframe view you want to modify is active.

2 From the **Options** menu, choose **View Options**.

Result: The *View Options* window is displayed.



Tip: You can also use the *View Options* tool on the *Standard* toolbar to open the view options.

3 Modify options as desired.

4 Click *OK*.

1 General

Options on the *General* tab affect the current scene.

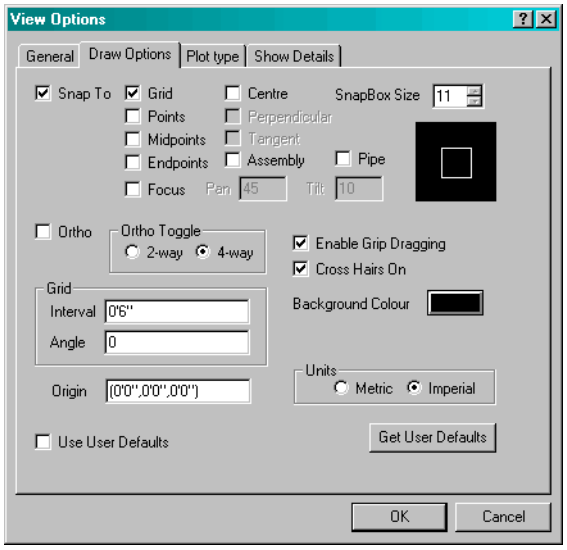
a. *Name*: name of the wireframe view.

b. *Follow User Scene*: select this check box to use the currently selected scene. Click to clear this check box and select the desired scene from the *Scenes* drop-down list. To learn more about scenes, refer to "Scenes" on page 6.5-15.

Tip: You can also change the current scene at any time by using the drop-down list on the *Scene* toolbar.

2 Draw Options

Options on the *Draw Options* tab affect the draw defaults. This tab is similar in function to the *Draw Defaults* tab available in User Options. For more information on the *Draw Defaults* tab, refer to "Draw Defaults" on page 5.2-13.

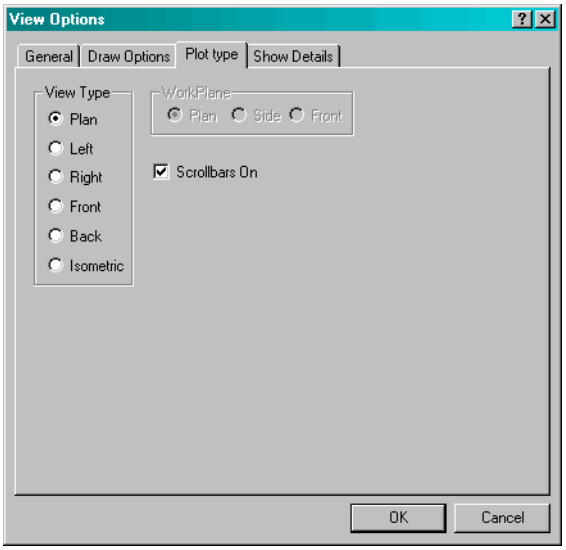


a. *Use User Defaults*: select this check box to reference the options selected on the *Draw Defaults* tab of *User Options*. Click to clear the *Use User Defaults* check box to make specific changes for the active view.

b. *Get User Defaults:* retrieves the values set on the *Draw Defaults* tab of *User Options*.

3 Plot Type

Options on the *Plot Type* tab affect the plot type of the active view.



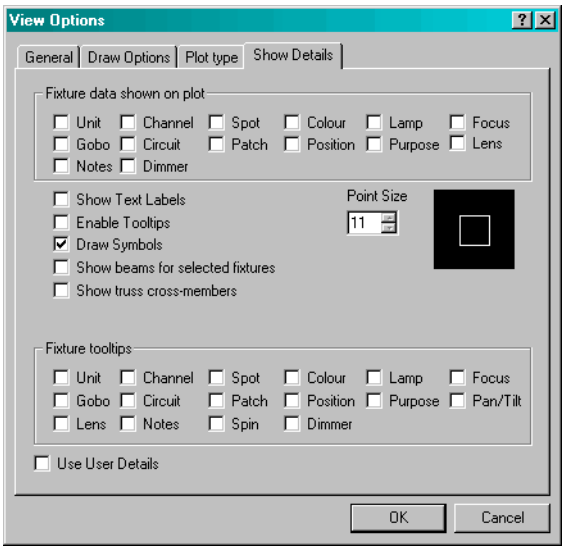
a. *View Type*: the view types available are as discussed in "Plot types" on page 6.2-4.

Tip: The view type can be toggled using the tools on the *CAD Options* toolbar.

b. *Scrollbars*: select this check box to display the scrollbars on the bottom and left side edges of the wireframe view. Click to clear this check box to turn the scrollbars off.

4 Show Details

Options on the *Show Details* tab affect what and how fixture, truss, text, and point information is displayed. This tab is similar in function to the *Show Details* tab available in User Options. For more information on the Show Details tab, refer to "Show Details" on page 5.2-14.



a. *Use User Details*: select this check box to reference the options selected on the *Show Details tab* of *User Options*. Click to clear the *Use User Defaults* check box to make specific changes for the active view.

4.6 Shaded Views

Shaded or open GL views offer a 3D perspective view of your drawing. Surfaces, truss and fixtures are “filled-in” or solid rather than outlined as in wireframe views.

Shaded views are not available in WYSIWYG Report.

Shaded views will follow Scenes just like any other view.

Keyboard and mouse control

To modify the point of view

- Use the arrow keys to move the position of the virtual target. PAGE UP/ PAGE DOWN will zoom you in and out.
- Holding down CTRL and using the arrow keys or the PAGE UP/PAGE DOWN will move the position of the virtual camera.
- At any time you can use the SHIFT key in combination with any of the movement keys to move in smaller increments.
- The mouse pan tool can be setup to rotate the model or the camera around the target. You can also use the mouse wheel to zoom in and out. The arrow keys will continue to move the target position as you do this.

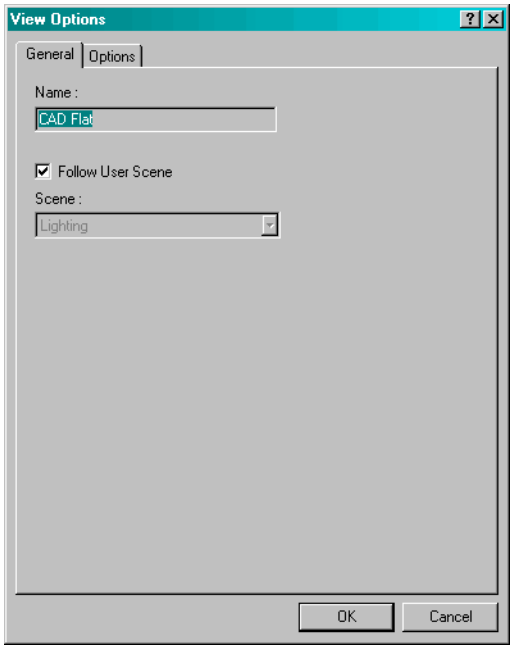
Modifying shaded views

The properties of a shaded view affect the view point, fixture beam quality, and ambient light. These settings are modified in the *View Options*.

To modify a shaded view

1 From the **Options** menu, choose **View Options**.

Result: The *View Options* window is displayed.



Tip: You can also use the *View Options* tool on the *Standard* toolbar to open the view options.

2 Modify options as desired.

3 Click *OK*.

1 General

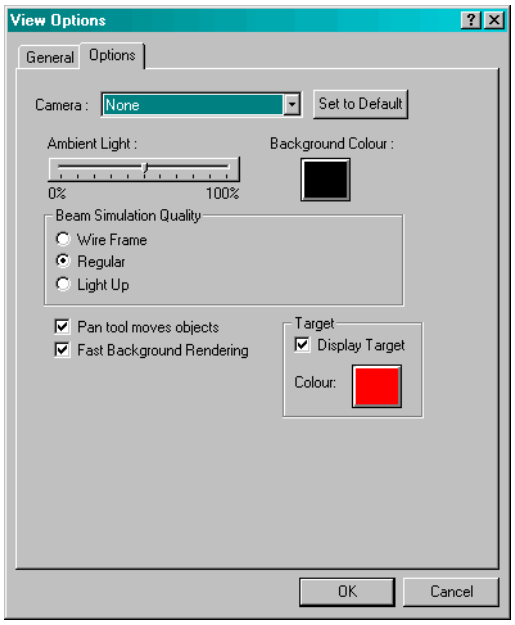
Options on the *General* tab affect the scene that is displayed in the shaded view.

a. *Follow User Scene*: select this check box to use the scene that is displayed in the drop-down list on the Scene toolbar. Click to clear the check box and select the desired scene from the *Scene* drop-down list. To learn more about scenes, refer to "Scenes" on page 6.5-15.

Tip: You can also change the current scene at any time by using the *Scene* toolbar.

2 Options

Options on the *Options* tab affect the view point, beam quality, and ambient light levels.



a. *Camera*: select a camera to change the point of view. Click *Set to Default* to remove any previous association with a camera and to display the original view point. This button is enabled if *None* is selected in the *Camera* drop-down list. You may need to click this button a few times to return the view point to its original setting. To learn more about cameras, please refer to "Drawing cameras" on page 6.4-9.

b. *Ambient Light:* use the slider to increase or decrease the amount of ambient light level shown in the shaded view.

c. *Background Colour:* modifies the background colour used in the shaded view. This colour is visible when you zoom far enough away that you can see beyond the venue.

d. *Beam simulation*: choose one of the following beam simulation qualities:

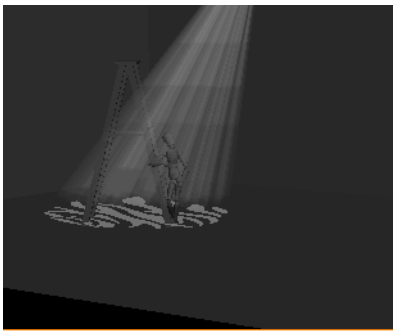
Wireframe. An outline of the footprint is shown and the beam is drawn based on the outline.

Regular. The true footprint is shown but objects are not illuminated.

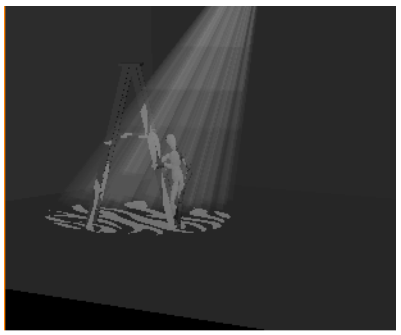
Lightup. Objects in the path of the beam are illuminated and a full footprint is shown.

The figures below illustrate the different levels of beam quality.

Wireframe



Regular



Light Up

e. *Display Target* : select the check box to show the focus point of the selected camera in the shaded view. Choose the colour of the displayed target by clicking the *Colour* box. The target is visible in shaded views only.

f. *Pan tool moves objects*: select this check box if you want to rotate the model on the target. Click to clear the check box if you want the pan tool to rotate the camera around the target.

You can use the following keys in addition to the mouse button to pan to the desired location:

Right - right arrow key

Left - left arrow key

Up - up arrow key

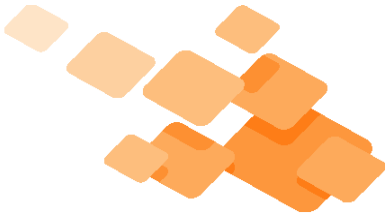
Down - down arrow key

You can also enable the Pan tool by choosing the Pan tool from the *View* toolbar.

g. *Fast Background Rendering*: select this check box if you want to render the beams and not the background. The background is displayed as a pasted image and the beams are rendered on top of the background.

Chapter 5.0

Managing Your WYSIWYG Document



This chapter provides information on how to work with your WYSIWYG document.

In this chapter

5.1 File Menu 5.1-1

5.2 Options 5.2-1

5.3 WYSIWYG Viewer 5.3-1

5.1 File Menu

The **File** menu contains the basic commands required to manipulate your WYSIWYG document.

The **File** menu is the same in all the modes.

New

The **New** command creates a new show document. If another show document is currently open, you will be prompted to save changes to that document before a new untitled file is created. Only one show document may be open at a time.

To create a new document

From the **File** menu, choose **New**.

Tip: You can also use the *New* tool on the *Standard* toolbar.

Open

The **Open** command opens a previously created show document. If another show document is currently open, you will be prompted to save changes to that document before another show document is opened. Only one show document may be open at a time.

To open an existing document

1 From the **File** menu, choose **Open**.

2 Select the file and then click *Open*.

You can open the following file types:

- WYSIWYG files (.wyg)
- WYSIWYG template (.wyt)
- WYSIWYG backup files (.bak)
- Wysiwygv2 files (.wys)
- DWG files (.dwg)
- DXF files (.dxf)
- LW2 files (.lw2)

DWG/DXF Export

This command exports the current show document to a DWG or DXF formatted file type. DWG and DXF formats are used to transfer documents to AutoCAD or other compatible drafting applications.

To export to DWG or DXF format

Refer to page “To export to DWG/DXF” on page 6.2-11.

Save

The **Save** command saves the open document to the same file name and location (folder, disk, etc.) as it was previously saved. If the document is being saved for the first time, this command will perform the Save As function.

To save your document

From the **File** menu, choose **Save**.

Tip: You can also use the *Save* tool on the *Standard* toolbar.

Save As

The **Save As** command is used to save the current document to a new file name, a new file type, or new destination, such as a floppy disk.

To save the current document

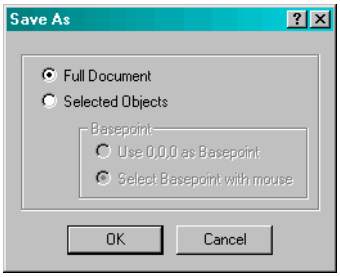
1. From the **File** menu, choose **Save As**.
2. In the dialog box that opens, use the browser to find the destination of the saved file. Ensure that the destination appears in the *Save in* box.
3. In the *File name* box, enter a name for the file.
4. In the *Save as type* box, choose the saved file type. There are a number of choices available, based on WYSIWYG versions.
5. Click *Save*.

WYSIWYG allows you to save an object or group of objects contained within your document as a separate file.

To save an object

1. Ensure the object(s) to be saved are selected.
2. From the **File** menu, choose **Save As**.

   Result: The following dialog box is displayed.



3. Select the *Full Document* radio button if you want to save the current document to a new file name, file type, or new destination. You are prompted to enter the destination and file name of the file.
4. Select the *Selected Objects* radio button if you want to save the selected object(s) as a separate WYSIWYG (.wyg) document.
5. Select the *Use 0,0,0 as Basepoint* radio button if you want to align the incoming drawing at 0,0,0 with the existing drawing.
6. Select the *Select Basepoint with mouse* radio button if you want to use the mouse to select the basepoint for the merge of the CAD drawings.
7. Click *OK*.

Template Files

The WYSIWYG **Save As** command lists a file type called .WYT. WYSIWYG template files are similar to the type of template file you may use on your word processor to set up fax cover sheets, memos or letters. Once you have drawn your venue with all of its lighting positions and everything else that is common to all shows, save it as a .WYT file in the \Templates directory. Each time you start a new drawing, that file will be available as a template to start from.

You can also enter all of you lighting equipment inventory into the *Flight Case* and save that as a template. When users start their drawing with your template, they will not have to choose from an endless list of fixtures but rather just what you have in stock.

This is also the ideal option for saving your preferred working settings. Template files save all of the shortcuts including custom reports, plots, etc. Toolbar settings however are not saved as they are saved by the application.

A list of most recently opened template files appears under the heading *Create New Plot* on the WYSIWYG welcome screen. The *more...* link will prompt you to open a template file from the WYSIWYG Templates directory.

To save a template file

1. From the **File** menu, choose **Save As**.
2. In the dialog box that opens, use the browser to find the destination of the saved file. Ensure that the destination appears in the *Save in* box. To ensure that your template file will be available on the WYSIWYG Welcome screen, save your file to the WYSIWYG/Templates directory.
3. In the *File name* box, enter a name for the file.
4. In the *Save as type* box, choose **WYSIWYG Template files (*.wyt)**.
5. Click *Save*.

Merge

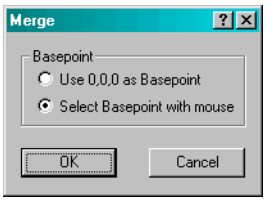
The **Merge** command is used to combine multiple show documents into one. All file types that can be opened directly by WYSIWYG can also be merged. Therefore an existing WYSIWYG document can be merged with:

- other WYSIWYG documents
- DWG/DXF files
- Lightwright2 files.

To merge documents

1. Open or create the destination document.
2. From the **File** menu, choose **Merge**.
3. In the dialog box that opens, choose the source file. You can change the types of visible files using the *File type* box.
4. Click *Open*.

   If CAD drawings are being merged, the *Merge* dialog box is displayed. Select the point that is used as the starting point for the merging of the two show documents.



5 Select the *Use 0,0,0 as Basepoint* radio button if you want to align the incoming drawing at 0,0,0 with the existing drawing.

6 Select the *Select Basepoint with mouse* radio button if you want to use the mouse to select the basepoint for the merge of the CAD drawings.

7 Click *OK*.

8 If you elected to select the basepoint using your mouse in step 6, you are required to click in the document or enter the coordinates to set the basepoint.

Close

The **Close** command closes the current show document and returns you to the *Welcome* window.

To close a show document

1 From the **File** menu, choose **Close**.

2 When prompted to save changes to the current show document, click *Yes* to save changes and close the document.

Click *No* to skip saving and close the document.

Click *Cancel* to skip saving and not close the document.

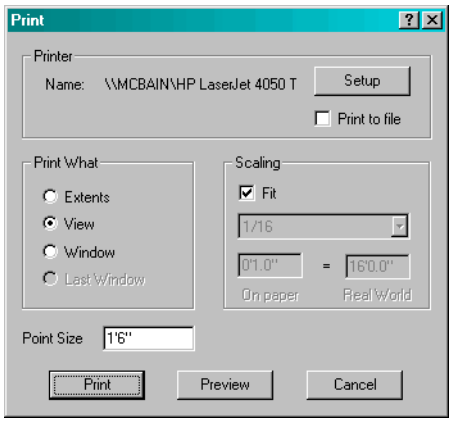
Print

The **Print** command varies from mode to mode. In general, this command prints the active view to the selected printer.

To print a document

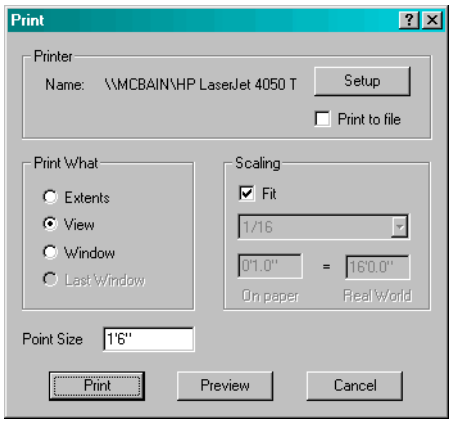
From the **File** menu, choose **Print**.

Depending on the mode you are in, this may print directly or it may open the following dialog box so that options can be set before printing can occur.



Considerations when printing from CAD mode

The **Print** command opens the *Print* dialog box.



This dialog box displays the printer that will be used to print your WYSIWYG document. The printer that is listed is the default printer you have set up in Windows. Click *Setup* to change the printer or printer settings.

Select the *Print to File* check box to print your WYSIWYG document to a file instead of to a printer. When you print, WYSIWYG prompts you to enter a file name and path.

The **Print** command will print the active window using the options you specify.

Print What options

- *Extents* prints the entire drawing.
- *View* prints the currently displayed view in the active window.
- *Window* allows you to draw a window around the area you wish to print.
- *Last Window* prints the portion contained in the last window you drew for a Window print. If you have not used or previewed a Window print yet, this option is unavailable.

Scaling options

- *Fit* will print the drawing to the scale required to fill the available print area based on the selection in the *Print what* group box. When the check box is selected, the drawing will print to fit.
- If the *Fit* check-box is unselected, you must choose the scale from the drop-down list. Standard scales are displayed in this menu. If the scale you want to use is not displayed, choose *Custom*.
- When *Custom* is selected, you must enter the ratio. Type the ratio in the *On Paper* and *Real World* boxes. The value in the *On Paper* box is the scale ratio that you want to use on a paper copy of the document. The value in the *Real World* box is the "real" measurement for the scale ratio that you specify in the *On Paper* box. For example, you may want to use a scaling of one inch in a paper copy of your document. This one inch measurement translates to a "realistic" measurement (for example, 3 feet).

The value in the *Point Size* box is the size at which points are printed.

While it is possible to print your drawing directly from the CAD mode, it is preferable to print plots from the Presentation mode. Presentation mode includes the ability to layout your plot with multiple views, reports, borders, and title blocks. Refer to "Plots" on page 9.3-1 for more information about printing plots.

Print Preview

Print Preview displays the document as it will look when printed. This allows you to double-check that the printed output is exactly what you want.

Print Setup

The *Print Setup* dialog box allows you to choose the print destination, the page orientation and size, and paper tray (if applicable). The print destination may be a physical printer, or it may be a file type or fax program, for example. Paper size and source options will correspond to the printer type selected.

Send

The **Send** command allows you to send your document to someone via e-mail. You must be connected to the Internet to use this feature. WYSIWYG will create a new message in your default mail program with the current WYSIWYG file attached.

Recent file shortcuts

The list of recent files at the bottom of the **File** menu gives you quick access to the last four files opened. This list will continually update as show documents are opened. If the file you are looking for is not on this list, use the **Open** command.

To open a recent file

1. From the **File** menu, choose the file you want to open.
2. If another show document is currently open, you will be prompted to save changes to that document before another show document is opened. Only one show document may be open at a time.

Exit

Use the **Exit** command to exit WYSIWYG. If you have made changes to your document but have not saved yet, you will be prompted to save your changes before exiting.

To exit WYSIWYG

1. From the **File** menu, choose **Exit**.
2. When prompted to save changes to the current show document, click *Yes* to save changes and exit WYSIWYG.

   Click *No* to skip saving and exit WYSIWYG.

Click *Cancel* to skip saving and not exit WYSIWYG.

5.2 Options

WYSIWYG offers you many options to customize the functionality of the software. Options are viewed and set in the **Options** menu.

The **Options** menu is the same in all the modes.

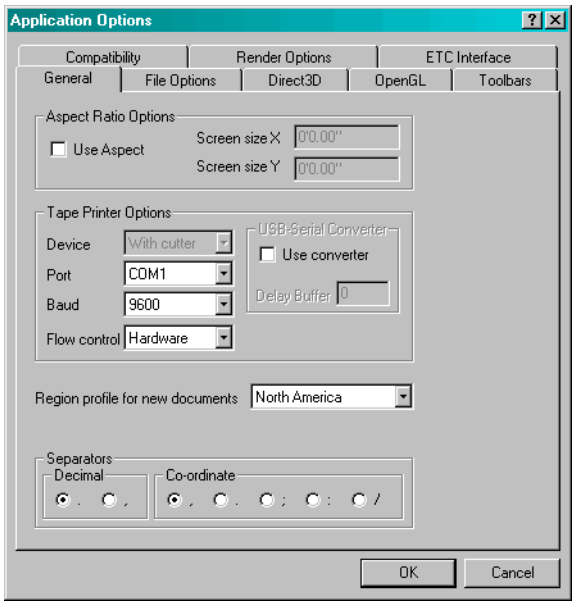
Application Options

Application Options are general settings for the WYSIWYG software and apply to the entire application regardless of the show file that you are working on. You might want to update these settings to display specific toolbars or indicate your file-saving preferences, for example.

To modify the application options

1 From the **Options** menu, choose **Application Options**.

Result: The *Application Options* dialog box is displayed.



1 General

Options on the *General* tab affect options for the pipe tape printer and the aspect ratio (height versus width).

a. *Use Aspect:* select this check box to specify an aspect ratio in the *Screen size X* and *Screen size Y* boxes.

b. *Screen Size X* and *Screen Size Y:* enter the size of the screen along the X and Y axis, respectively.

c. *Device*: specify whether or not the pipe tape printer has a tape cutter.

d. *Port* : specify which port on your computer the pipe tape printer is connected to.

e. *Baud* : specify the baud rate at which the pipe tape printer is running. The baud rate is the rate at which your modem can transmit and receive data.

f. *Flow Control* : select the method of data control between the pipe tape printer and the machine. Available options are Hardware, Software, and None. The default is Hardware.

g. *Use converter*: select this check box to indicate that a USB Serial converter is to be used.

h. *Delay Buffer*: enter the buffer size for the USB Serial converter.

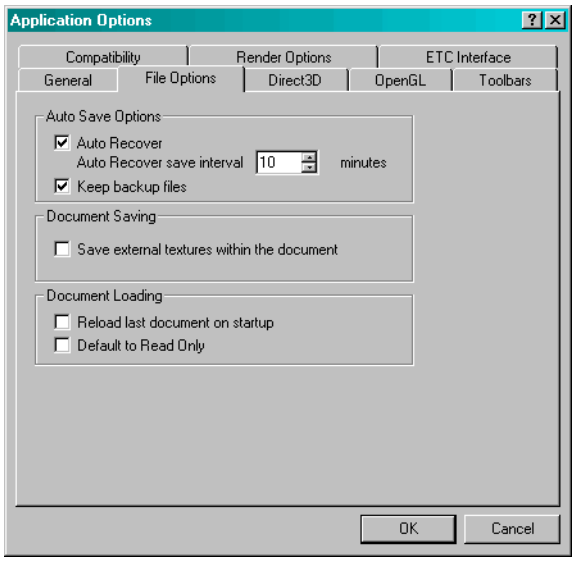
i. *Region profile for new documents*: the profile to be used for such items as voltage and which default lamp should be used in a fixture.

j. *Decimal*: select the method you want to use to separate whole numbers from decimals. Available options are comma and period. Note that the decimal separator cannot be the same as the co-ordinate separator.

k. *Co-ordinate*: select the method you want to use to separate coordinates in WYSIWYG. Available options are comma, period, semicolon, colon, or slash. Note that the co-ordinate separator cannot be the same as the decimal separator.

2 File Options

Options on the *File Options* tab affect your file saving settings. It is important to save your work often during your work session. WYSIWYG has an Auto Save function that will help to keep your data safe, however, it is always a good idea to get in the habit of saving your show document to an external disk periodically during your work session.



a. *Auto Recover*: select this check box to save your document at the intervals specified in the *Auto Recover save interval* box. If for some reason WYSIWYG closes unexpectedly, such as in the case of a power loss, your document can be recovered. This recovery will only be up-to-date as of the last Auto Save, so it is recommended that you do not set this interval to be too long.

b. *Keep Backup Files*: select this check box to save a copy of your document with the ".bak" extension every time you save the file. The backup copy is one save behind the saved copy and is saved in the same folder as the original document. Each new backup copy replaces the previous

backup copy. If the main show document becomes corrupted, you can open the backup file to recover your data.

The default settings for these options are *Auto Recover* enabled with an interval of 10 minutes and *Keep Backup Files* enabled.

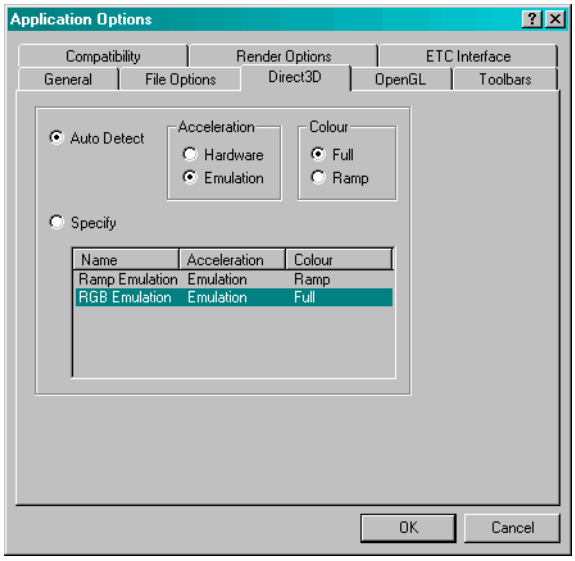
c. *Save external textures within the document*: select this check box if you want the jpeg and/or bitmap used as textures to be saved within your WYSIWYG document. The textures will "travel" with the document and will be visible regardless of the computer on which the document is opened.

d. *Reload last document on startup:* select this check box to ensure that the last document that you have been working on in WYSIWYG is loaded automatically when you launch WYSIWYG.

e. *Default to Read Only*: select this check box to load documents in read-only format. This format is helpful if you are loading someone else's document and you do not want to accidentally make changes to it.

3 Direct 3D

Options on the *Direct 3D* tab affect the setting of the DirectX components to be used for graphics. You can select these components manually or have them automatically detected. By default, WYSIWYG uses Open GL graphics technology in all simulation views. If you want, you can set WYSIWYG to use Direct 3D instead of Open GL, in which case the following options are available.



a. *Auto Detect*: WYSIWYG attempts to find a DirectX driver that corresponds to the options you have set for *Acceleration* and *Colour.* The driver that it chooses is highlighted in the list of available drivers below.

b. *Acceleration*: enables either hardware or software (emulation) acceleration.

Hardware acceleration uses the capabilities of the graphics hardware of the machine to display graphics. This is the default option. Emulation uses the emulation library or software components to display graphics.

c. *Colour:* enables either full or ramp colour.

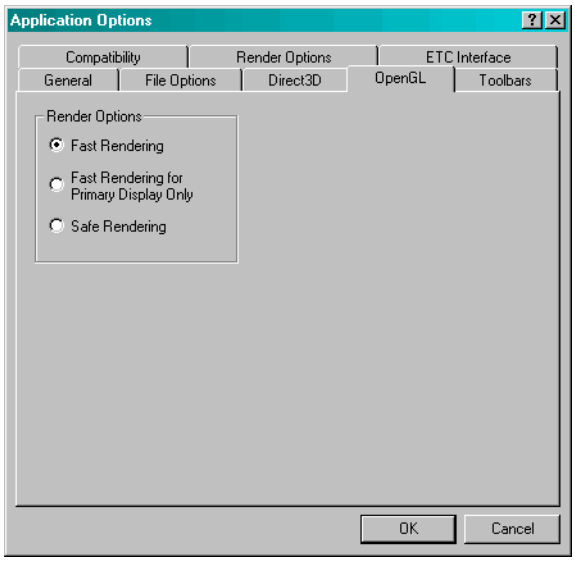
Full displays the full colour content of the fixture sources and the material of the object being lit. This is the default option.

Ramp displays only the gray component of each fixture specified in a scene. The colour components of the fixtures are ignored.

You can also manually select a DirectX driver from the list of drivers. The list changes based on current video card and mode.

4 Open GL

Options on the *Open GL* tab affect the type of rendering that you want to use.



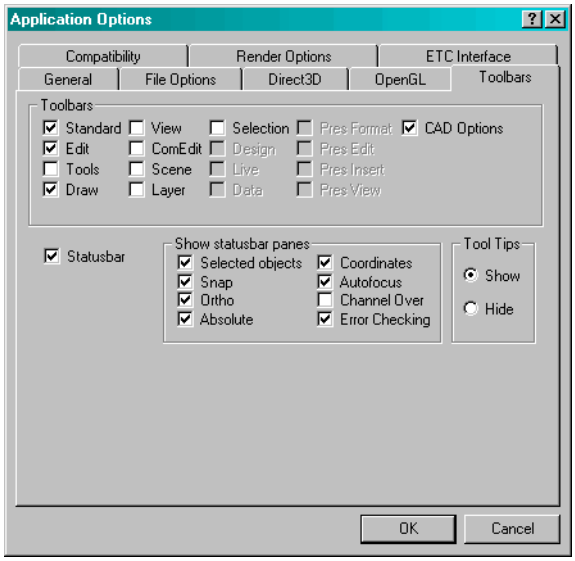
a. *Fast rendering:* enables hardware acceleration. Hardware acceleration uses the capabilities of the graphics hardware of the machine to display graphics. This is the default option.

b. *Fast rendering for primary display only*: enables hardware acceleration for the primary emulation.

c. *Safe rendering*: enables software emulation.

5 Toolbars

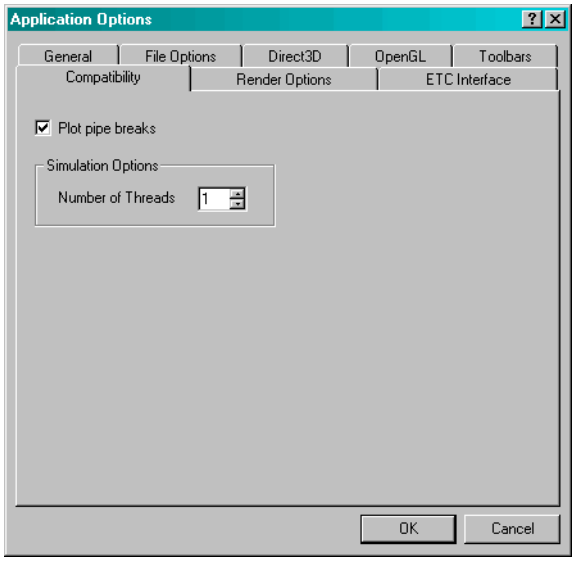
Options on the *Toolbars* tab affect which toolbars and status bar panes you want to display in WYSIWYG.



You can also hide the tooltips that display information about a specific fixture when you hover over the fixture. Tooltips are displayed by default. For more information on toolbars, refer to “Toolbars” on page 4.3-1.

6 Compatibility

Options on the *Compatibility* tab affect the processing power for beam simulation.

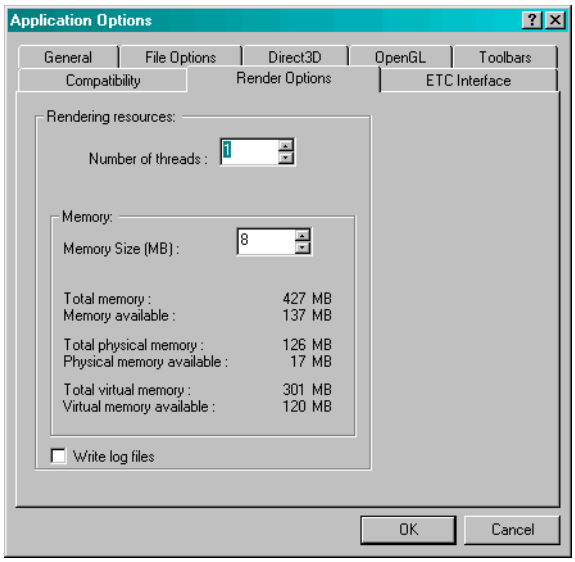


a. *Plot pipe breaks:* select this check box to plot pipe breaks.

b. *Number of threads*: number of processors you have available on your Windows machine for beam simulation. Modify this number only if you upgrade the number of processors.

7 Render options

Options on the *Render Options* tab affect the available memory and processing power available for rendering.



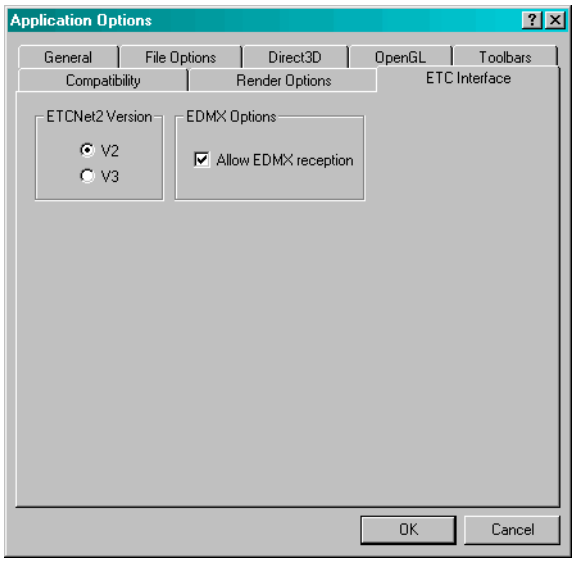
a. *Number of threads*: number of processors available for rendering. Modify this number only if you upgrade the number of processors on your machine.

b. *Memory Size:* amount of memory the renderer can use for temporary files. 8MB is the default and should be sufficient for most machines. If you are running many applications, you may want to decrease this number, or if your machine has a lot of RAM, you may want to increase this number.

c. *Write log files:* select this check box to generate a log file during rendering.

8 ETC Interface

Options on the *ETC Interface* tab affect the options that will allow you to monitor EDMX data directly. Direct EDMX reception allows WYSIWYG Perform users to monitor EDMX data without the need to connect to a console using the Device Manager.



a. *ETCNet2Version*: select which ETCNet2 network version you want to monitor. You can monitor only one version at a time.

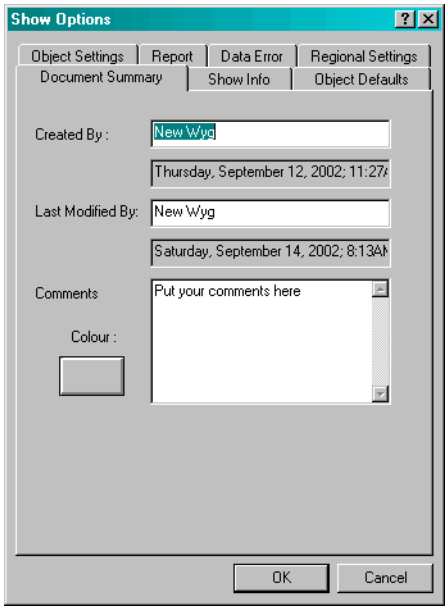
b. *Allow EDMX reception*: select this check box to enable direct EDMX reception in WYSIWYG.

Show Options

Show settings that are specific to the current show file, such as show name, designer, and venue, are set here. You might want to modify these settings when you start a new show file. Error conditions and how certain objects are displayed in your show file are also specified in Show Options.

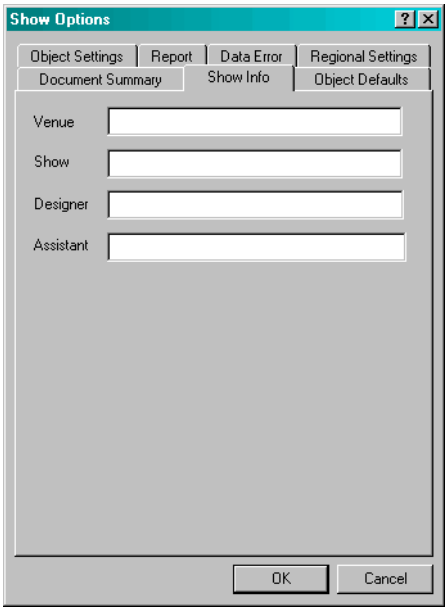
1 Document Summary

Options on the *Document Summary* tab affect the document information (such as name of the person who created the document and the name of the person who last modified the document). The date stamps are not editable, but will track the creation and modification dates of the file.



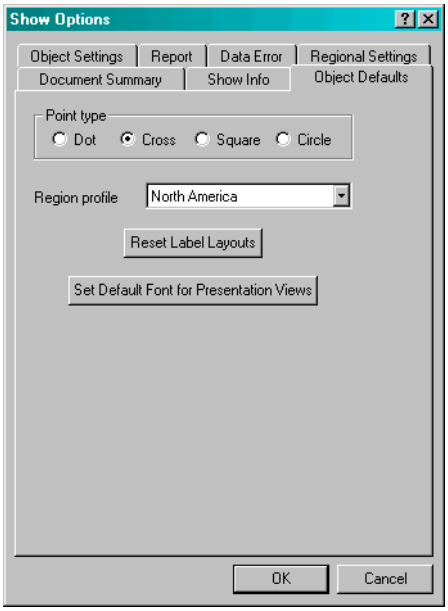
2 Show Info

Options on the *Show Info* tab affect the venue, show, Designer, and Assistant's names that are displayed in your show document.



3 Object Defaults

Options on the *Object Defaults* tab affect the default settings for the regional profile. The profile that you choose determines items such as the type of bulb to use in a particular fixture. This also sets how points will be displayed (as crosses, dots, squares, or circles).



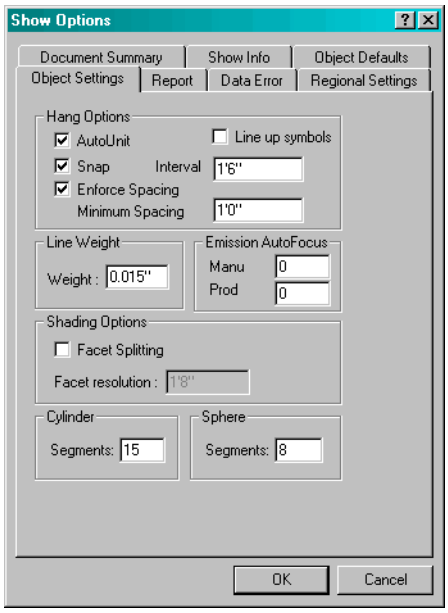
a. *Point type:* indicates the default point type in your show document.

b. *Reset Label Layouts:* click to reset all fixture labelling to its default settings.

c. *Set Default Font for Presentation Views*: click to select a default font for presentation views.

4 Object Settings

Options on the *Object Settings* tab affect how various objects are placed or drawn in CAD mode.



a. *AutoUnit:* select this check box to automatically assign a unit number to each fixture that is hung. This option is available for pipes only. It does not apply to truss.

b. *Snap*: select this check box to enable pipe snap. Pipe snap places fixtures on a pipe at a specific interval as specified in the *Interval* box. For information on snaps, refer to "Pipe snap" on page 6.8-2.

c. *Enforce spacing*: select this check box to indicate that the spacing you specify in the *Minimum Spacing* box is enforced.

d. *Minimum spacing:* enter the minimum spacing between fixtures. This option prevents you from hanging the fixtures too close to one another. The minimum spacing cannot be greater than the value you specify in the *Snap Interval* box.

e. *Line up symbols:* select this check box to line up fixtures at 45 degree increments. This setting does not affect the focus of the beam.

f. *Interval*: enter the distance between fixtures. This option is used for pipe snaps.

g. *Weight*: enter the default weight of a line when it is printed or viewed using print preview.

h. *Facet Splitting:* select this check box to enable facet splitting in shaded views. Facet splitting splits objects in a plot into small triangles for a smooth beam fallout during simulation in a shaded view.

i. *Facet Resolution*: enter the size of the triangles. The more triangles you have the more realistic the cosine distribution will be. However, this also decreases the simulation speed. Note that this is applicable to Direct 3D only.

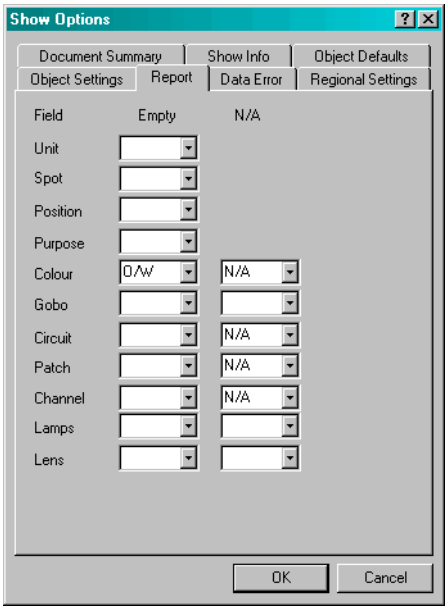
j. *Manu*: enter the manufacturer's code for those fixtures that are Autofocus-compatible.

k. *Prod*: enter the product code for those fixtures that are Autofocus-compatible.

l. *Segments*: enter the number of segments into which you want to break a cylinder or sphere.

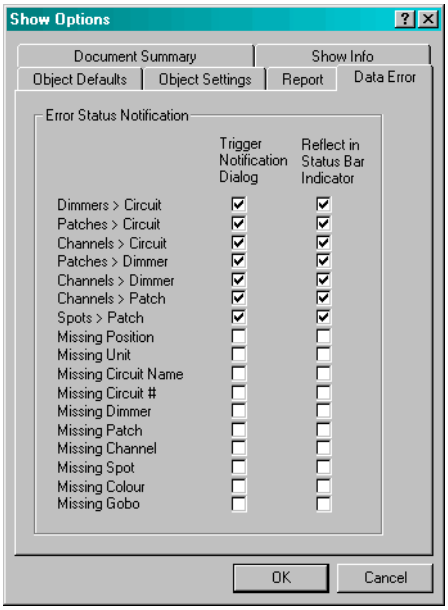
5 Report

Options on the *Report* tab affect the default values for empty or non/applicable cells in spreadsheets.



6 Data Error

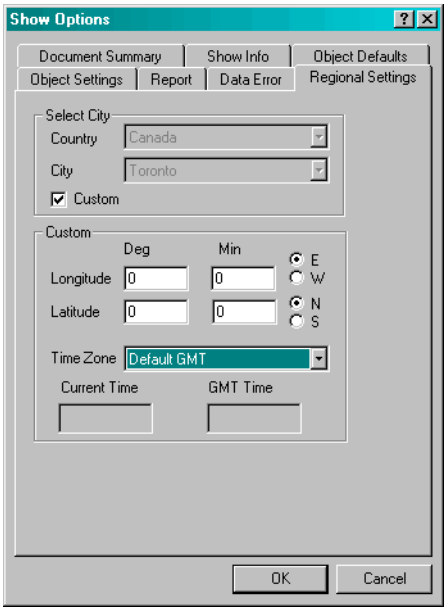
Options on the *Data Error* tab control what types of data errors trigger notification boxes and/or status bar notifications.



For more information on data errors, refer to “Errors” on page 7.4-1.

7 Regional settings

Options on the *Regional settings* tab set the geographic location of your show. You can either select the country and city or enter the latitude and longitude of the location. These settings are primarily used for renderings that use environmental options.



a. *Custom*: select this check box to enter the latitude and longitude values in the *Latitude* and *Longitude* boxes, respectively. Click to clear this check box to select a country and city from the *Country* and *City* drop-down lists, respectively.

b. *Time Zone*: select the time zone for your show. When you select a time zone, the current time as set on your computer is displayed in the *Current Time* box. The GMT value is also shown.

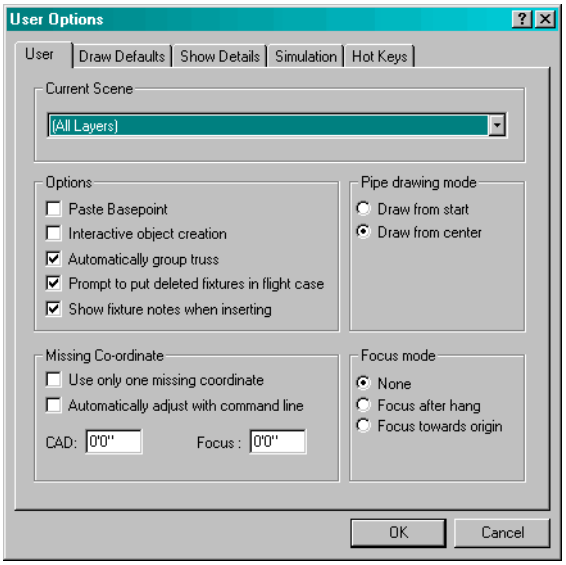
The values that you enter on this tab are reflected in Step 6 of the Render Wizard. For more information on the Render Wizard, refer to "Rendering" on page 8.6-1.

User Options

User Options are available so that you may set up your working preferences. You might, for example, want to specify how the snap operation will work or how much information will be displayed on your plot. Settings that are established here are in effect for the entire time that you are working in WYSIWYG regardless of the show file that you are working on.

1 User settings

Options on the *User* tab affect the current scene selection, options, pipe drawing mode, missing co-ordinate, and focus mode settings.



a. *Paste Basepoint*. select this check box to enter a basepoint before an object is copied or cut. This will be the object's reference point for paste commands. Click to clear this check box to allow the basepoint to be the insertion point of the object.

b. *Interactive object creation:* select this check box to allow you to draw objects using the mouse instead of entering values in dialog boxes.

c. *Automatically group truss:* select this check box to group truss objects together in a similar manner as choosing **Group** from the **Edit** menu. This option is helpful in that it treats the truss as a single unit and allows you to perform actions more effectively on multiple objects. If you want to edit one truss object, you must use the Ungroup command to remove the grouping from the truss objects.

d. *Prompt to put deleted fixtures in flight case*. select this check box to be prompted when you delete a fixture. You will be prompted to delete it completely or send it to the Flight Case.

e. *Show fixture notes when inserting*. select this check box to display the fixture notes when you are hanging fixtures, if applicable to the selected fixture.

f. *Pipe drawing mode:* sets the preference for drawing pipes (either from the start or from the center). For more information on drawing pipes, refer to "Drawing pipes" on page 6.7-3.

g. *Missing coordinate*: sets missing coordinate options. For more information on missing coordinates, refer to "The missing coordinate" on page 6.2-3.

h. *Focus mode:* allows you to choose to focus a fixture as you hang it in your drawing. Specify one of the following options to set how a fixture will react after you hang it in a plot:

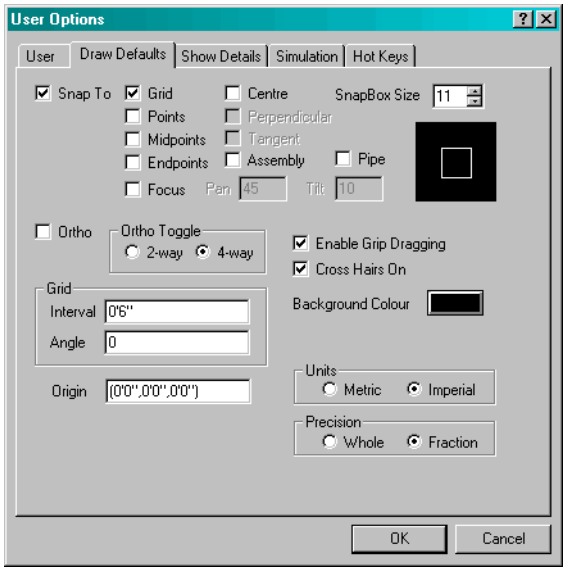
None. This setting indicates that you can continue to hang fixtures uninterrupted.

Focus after hang. If enabled, you will be prompted to focus each fixture as you hang it. Once the fixture is focused, you can continue hanging other fixtures.

Focus towards origin. If enabled, the fixtures that you hang are automatically focused towards the user origin as you hang them. You can continue to hang other fixtures uninterrupted.

2 Draw Defaults

Options on the *Draw Defaults* tab affect general preferences for snap, ortho, grid and units settings.



a. *Snap To*: enables or disables snap. For more information on snaps, refer to "Snaps" on page 6.2-6. Snap tools can be toggled at any time using the tools on the CAD Options toolbar.

b. *SnapBox Size:* sets the size of the snap box that appears around the cursor when in snap mode. Consequently, this affects how close the cursor must be to the objects before the snap is applied.

c. *Ortho Toggle*: sets either 2-way or 4-way ortho settings. For more information on these settings, refer to "Ortho" on page 6.2-5. The Ortho settings can be toggled at any time using the tools on the CAD Options toolbar.

d. *Enable Grip Dragging:* select this check box if you want to be able to resize objects by clicking and dragging on their markers.

e. *Cross Hairs On:* select this check box to set the cursor display to use cross hairs in plot views. Cross hairs are useful for lining up objects.

f. *Background Colour:* the colour that will be used in all wireframe views.

g. *Interval*: the spacing between points on the drawing grid. Although the WYSIWYG drawing grid might not be visible, it exists in the background. The settings in this group box are important when using the snap to grid tool. For more information on this tool, refer to "Snaps" on page 6.2-6.

h. *Angle*: angle of rotation for the drawing grid. Although the WYSIWYG drawing grid might not be visible, it exists in the background. The settings in this group box are important when using the snap to grid tool. For more information on this tool, refer to "Snaps" on page 6.2-6.

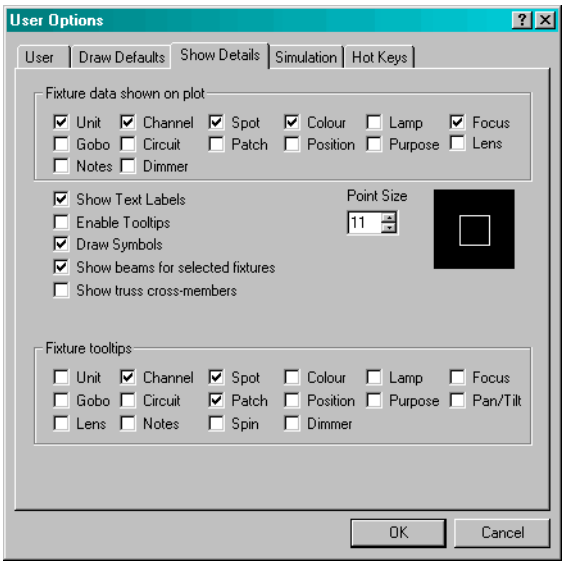
i. *Origin*: point of origin. For more information on the origin, refer to "Coordinate system and origin" on page 6.2-1.

j. *Units of measure*: for more information on these units, refer to "Metric vs. Imperial" on page 6.2-2. The units can be toggled from metric to imperial at any time by double clicking on the coordinates display on the status bar.

k. *Precision*: affects the units of drawing precision. For more information on these units, refer to "Coordinate notation" on page 6.2-1. Based on your choice for measurement units, you can choose to display the drawing precision in centimetres, millimetres, whole numbers, or fractions.

3 Show Details

Options on the *Show Details* tab affect the amount of information that is shown on your drawing. This includes how much data is shown on the plot.



a. *Fixture data shown on plot:* indicates the fixture data that you want to display on your drawing.

b. *Show Text Labels*: select this check box to display text labels.

c. *Tooltips:* displays tooltips on your drawing. Tooltips are the pop-up messages that display specific information about a fixture when you hover over the fixture. Choose the information you want to display in a tooltip by selecting the appropriate check boxes in the *Fixture tooltips* group box. Tooltips are often helpful when trying to locate a specific fixture in a plot.

d. *Draw Symbols:* select this check box to draw fixtures as symbols rather than as 3D objects. For more information on symbols, refer to "Symbols" on page 6.8-16.

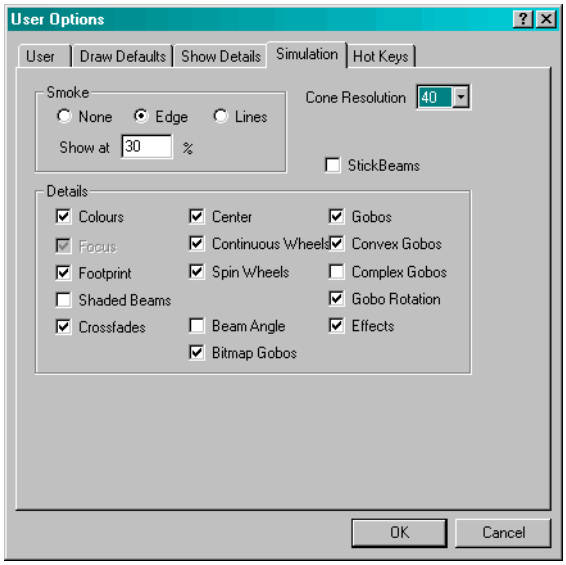
e. *Show beams for selected fixtures:* select this check box to automatically turn on beams when selecting a fixture on your drawing. If you do not select this box, the beam is not visible when you select a fixture.

f. *Show truss cross members*: click to clear this check box to hide truss cross members on your drawing to reduce the clutter or increase the speed of the display. Select this check box to see the cross members.

g. *Point size:* default size for all points is set in the box. You can view a sample of the chosen size in the box to the right.

4 Simulation

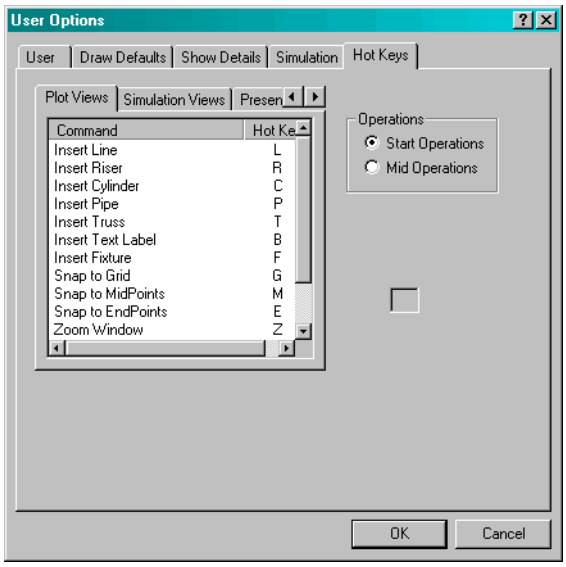
Options on the *Simulation* tab are only set for WYSIWYG Design and Perform. These settings determine how detailed simulation views are, and can affect the speed of beam simulation in simulation views.



For more information on simulation options, refer to “Simulation Options” on page 10.5-1.

5 Hot Keys

Options on the *Hot Keys* tab allow you to assign hot keys to common functions.



Hot keys speed access to commonly used commands. For example, when you are working in a wireframe view in CAD mode, you can insert a riser by pressing the R key on your keyboard, instead of choosing **Riser** from the **Draw** menu.

You can change the hot key settings by clicking the command you want to change and entering the new hot key into the box to the right of the list. Tabs above the hot key list allow you to view the hot key settings for the different views.

a. *Start Operations*: views the commands that are available before you create an object or any time a view is in its idle state.

b. *Mid operations*: views the commands that are available during the midst of an operation.

View Options

View options set how objects are drawn and displayed in Wireframe and shaded views.

For more information on View options for Wireframe views, refer to "Modifying wireframe views" on page 4.5-2.

For more information on View options for shaded views, refer to "Modifying shaded views" on page 4.6-1.

5.3 WYSIWYG Viewer

The WYSIWYG Viewer is a separate application that allows non-WYSIWYG users to view and print your WYSIWYG (.wyg) files. It is beneficial to those individuals who are interested in viewing or printing a WYSIWYG file but who do not own a licensed copy of WYSIWYG.

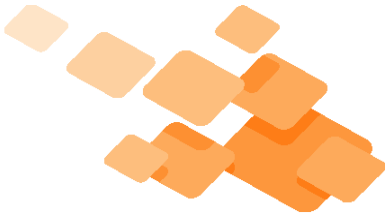
Any file with a .wyg file extension can be viewed and printed from the WYSIWYG Viewer.

Presentation mode is the only mode available in the WYSIWYG Viewer. All functionality except for navigation and printing has been disabled. Ensure that all relevant information is available in this mode before your file is opened in the WYSIWYG Viewer. Files cannot be edited and items cannot be inserted into a file. For more information on preparing your file in Presentation mode, refer to Chapter 9.0, “Presentation Mode.”

The WYSIWYG Viewer can be downloaded from the www.cast-soft.com Web site.

Chapter 6.0

CAD Mode



CAD mode is where you create your show drawings. This includes drawing your venue, set pieces, lighting positions, focus positions and lighting fixtures. CAD mode operates like many other CAD programs, so many of the concepts will be familiar to those who have used a computer-aided drafting program before. WYSIWYG adds features that are specific to the entertainment industry, such as a comprehensive 3D library containing truss, lighting equipment, lighting accessories, props, musical instruments, and various human figures.

In this chapter

6.1 Layout Tabs	6.1-1
6.2 The CAD Environment	6.2-1
6.3 The Library Browser	6.3-1
6.4 Drawing Objects	6.4-1
6.5 Editing Objects	6.5-1
6.6 CAD Tools	6.6-1
6.7 Hang Structures	6.7-1
6.8 Hanging and Focusing Fixtures	6.8-1
6.9 Colour, Gobos, and Accessories	6.9-1
6.10 The Flight Case	6.10-1
6.11 Rendering	6.11-1

6.1 Layout Tabs

Beneath the work area in each mode is a series of layout tabs. These layouts provide various configurations of the views you are working with. To change layouts, click the tab that corresponds to the layout you wish to use. The CAD mode contains the following layouts:

- Wireframe - work area displays a full screen wireframe view.
- Quad - work area is divided into quadrants, three of which can be modified to show plan, front, side, or isometric wireframe views. The lower-right quadrant contains a shaded view.
- Flight Case - Flight Case is displayed in a section of the work area along with a wireframe view.
- Shaded - work area displays full screen shaded view.

6.2 The CAD Environment

Scales

When drawing in WYSIWYG, you are drawing in real scale (1:1). When you create a drawing in CAD, you are generating a virtual representation of your real set-up. If you were to do this on paper by hand, you would need to draw a scaled-down version of your space. Because in the CAD mode there are no paper size limitations, you can draw your venue, sets, pipes, trusses and lighting fixtures in real scale.

Scaling down for printing purposes is done during print set-up and in the Presentation mode when creating plots. These settings allow you to print your drawings in whatever scales are necessary without having to redraw anything.

Coordinate system and origin

When working in CAD you are working in a 3D environment (even in WYSIWYG Report). Objects are drawn as 3D objects, with width, depth and height values using the Cartesian coordinate system of 3 working axes X, Y, and Z. The point where the 3 axes meet is called the origin and the value of X, Y, and Z is 0 respectively (0,0,0).

By default the origin is set at the center point of the WYSIWYG venue that you insert. Inserting venues is discussed in "Drawing a venue" on page 6.4-1. The origin can be reset to another point in your drawing; you can set a user origin whereby a different point will assume the values (0,0,0).

To set the user origin

1. From the **Tools** menu, choose **Set User Origin**.

   Tip: You can also use the *Origin* tool on the *Tools* toolbar.
2. Click a point on your drawing or type in the coordinates (X,Y,Z) that will assume the values 0,0,0.

To reset the user origin

From the **Tools** menu, choose **Reset User Origin.**

Result: This will reset the origin back to the WYSIWYG default origin.

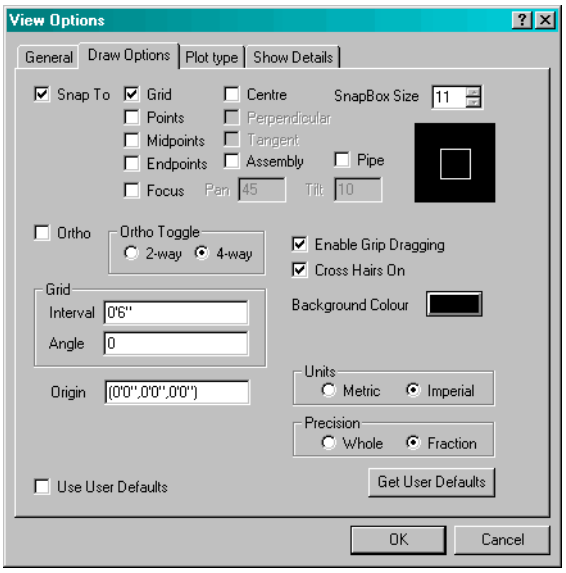
Coordinate notation

When entering coordinates in WYSIWYG, you can specify a location or distance in either inch fractions (to the sixteenth of an inch) or millimetres. For example, you can specify a measurement of 1'6"3/16", which translates to 1 foot, 6 and 3/16 inches.

To set default options

1. From the **Options** menu, choose **View Options**.
2. Click the *Draw Options* tab.

3 Clear the *Use User Defaults* check box to modify the default setting for *Precision*.



4 Based on your choice for measurement units, you can choose to display the drawing resolution in whole numbers, fractions, centimetres, or millimetres.

Whole - indicates that the measurements are rounded and displayed to the nearest whole number.

Fraction - indicates that the measurements are displayed to the nearest sixteenths of an inch.

cm - indicates that the measurements are displayed in centimetres.

mm - indicates that the measurements are displayed in millimetres.

Metric vs. Imperial

WYSIWYG allows you to use metric or imperial units at any time. You may choose to set a default type of unit, and you may choose to switch unit type on the fly.

You can also indicate whether you want these units measured in whole numbers, fractions (to the sixteenth of a inch), centimetres, or millimetres.

To set the default unit type

1 From the **Options** menu, choose **User Options**.

2 Click the *Draw Defaults* tab.

3 Click either *Metric* or *Imperial*.

4 Click *Whole, Fraction, mm* or *cm*. These choices vary depending on whether you selected Metric or Imperial in step 3.

To switch unit types on the fly

Double-click on the coordinates display on the status bar.

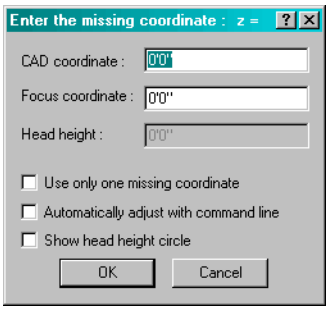
The missing coordinate

The missing coordinate (X,Y,or Z) is the coordinate whose value cannot be entered by clicking on the screen. The easy way to determine which coordinate is the missing coordinate in a wireframe view is to move the mouse around. Look at the status bar at the bottom of the working area. You will see only two values changing. The value that is not changing is the missing coordinate for that view or workplane. The missing coordinate is dependent on the plot type and the workplane selected. In the following example, Y is the missing coordinate as its value is set at 0’0”.

Once entered this value will affect all subsequent objects inserted in the current view. For example, if the missing coordinate is set to 5’ in a plan view, all objects will be placed 5’ off the floor (X,Y,5) until the missing coordinate is changed again. Please note that the missing coordinate is not a move tool. Objects will not be moved to the missing coordinate value. The missing coordinate only affects subsequent inserts.

To enter a value for the missing coordinate

1 Press the TAB key on your keyboard or click the *Missing Coordinate* tool on the *CAD Options* toolbar.



2 Enter the value for the CAD coordinate in the *CAD coordinate* box.

3 If you want to focus fixtures while inserting objects, enter a value for the focus coordinate in the *Focus coordinate* box.

4 To check the light coverage at a certain height (the height of the performer), enter a value in the *Head height* box.

Note: This box is enabled when the *Show head height circle* check box is selected.

5 If you want to use only the CAD coordinate for focusing and placing/editing objects, select the *Use only one missing coordinate* check box.

6 To automatically adjust the missing coordinate with any 3D coordinate value that you enter on the command line, select the *Automatically adjust with command line* check box.

7 Select the *Show head height* check box to display the coverage of beams at the head height selected in the *Head height* box.

8 When you are finished with your selections, click *OK*.

Example

1. Change to the plan view in your drawing.
2. Press the TAB key and enter a value of 0" for the missing CAD coordinate.
3. Insert a few objects on the stage. All of these objects are drawn resting on the stage.
4. Change the missing CAD coordinate to a height of 10’.
5. Insert a riser.
6. Although we are in a plan view the riser has been drawn at the height of 10' specified in the missing coordinate dialog box.
7. Change to a side view and you will see that the riser has been placed above the stage.

Tips:

- You can use separate missing coordinates for focusing fixtures if you enter a value for the Focus coordinate when pressing TAB. The advantage of this is you can be adding pipes and other CAD elements at the specified missing CAD coordinate, but still focus the fixtures at an elevation of 5’ for example.
- If you enter a value in the *Head height* field and select the *Show head height circle* check box, all subsequently focused fixture’s beams will display both the coverage at the specified head height as well as the footprint of the beam on the surface below.

Plot types

All objects occupy three-dimensional space. Different plot view types allow you to see and work with your drawing from different perspectives.

There are six types of plot views, accessible from the CAD Options toolbar. The following definitions identify the working axes X, Y, and Z and the missing coordinate for each plot type. The missing coordinate can be defined as the axis for which a value cannot be set simply by clicking on the work space. Refer to “The missing coordinate” on page 6.2-3 for more information on the missing coordinate.

Plan View

Plan views display the plot from above looking down. This is similar to a plan view drawing on paper. In plan views the working axes are X and Y and the missing coordinate is Z.

Left View

Left views display the plot looking from the left side through the venue. This is similar to a section on paper. In left views the working axes are Y and Z and the missing coordinate is X.

Right View

Right views display the plot looking from the right side through the venue. This is similar to a section on paper. In right views the working axes are Y and Z and the missing coordinate is X.

Front View

Front views display the plot looking from the front side through the venue. This is similar to an elevation on paper. In front views the working axes are X and Z and the missing coordinate is Y.

Back View

Back views display the plot looking from the back side through the venue. This is similar to an elevation on paper. In back views the working axes are X and Z and the missing coordinate is Y.

Isometric View

An Isometric view is a 3D perspective drawing. Despite this you are still limited to two working axes. In isometric views the working axes and the missing coordinate are dependent of the workplane selected. The workplanes available are:

- plan
- side
- front

If a plan workplane is selected, the working axes are X and Y and the missing coordinate is Z.

If a side workplane is selected, the working axes are Y and Z and the missing coordinate is X.

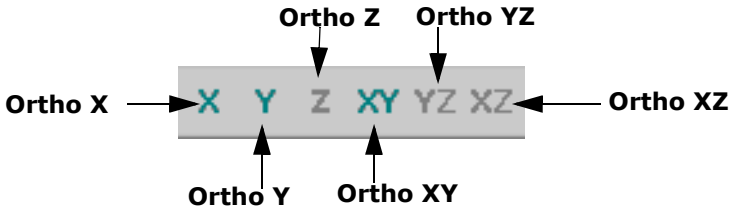
If a front workplane is selected, the working axes are X and Z and the missing coordinate is Y.

The crosshairs of your cursor will change to reflect the selected workplane.

Note: Isometric views are not available in WYSIWYG Report.

Ortho

When ortho mode is inactive, objects can be drawn or moved in any direction on the workplane. You can draw diagonal lines or move objects anywhere in the working area. Ortho mode constrains movement and drawing to a direction parallel to the specified axis.



Ortho can be activated at any time by clicking the ortho tools on the *CAD Options* toolbar, using the **Ortho** command on the right-click menu, or by using the F8 key on the keyboard. The ortho mode(s) available will depend on two things:

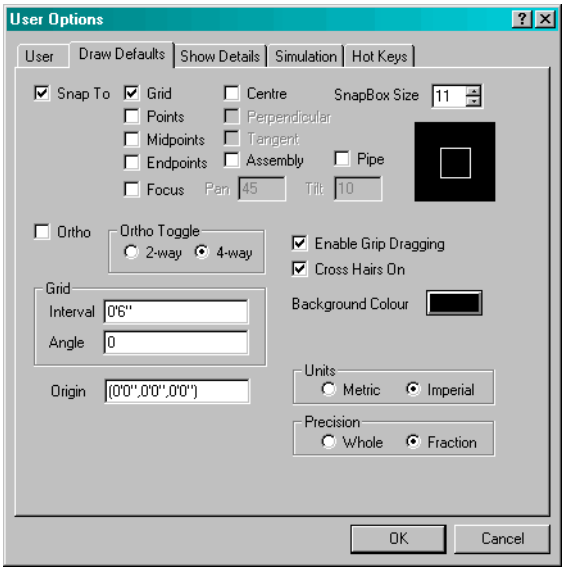
- The plot type determines the axes of movement; XY for plan views, YZ for side views, XZ for front and back views.
- You have the option of enabling two-way or four-way ortho.

In two-way, you always have the option of moving parallel to both axes (XY, YZ, XZ). In four-way, you can also constrain to only one axis, (for example, X only, Y only, or XY).

To set two-way or four-way toggle

1 From the **Options** menu, choose **User Options**.

2 Click the *Draw Defaults* tab.



3 Click either *2-way* or *4-way*.

2-way mode toggles between off and combined mode. For example, in plan view 2-way toggle, F8 toggles between Ortho off and Ortho X-Y.

4-way mode toggles between off, combined, and single modes. For example, in plan view 4-way toggle, F8 toggles between (Ortho off), (OrthoX), (Ortho Y) and (Ortho X-Y).

4 Click *OK*.

Snaps

Snaps are used to assist in the drawing and placement of objects. When a snap setting is active, the cursor will be drawn to the applicable snap point. You can have multiple snaps active at the same time. Snap selections can be made on the CAD Options toolbar or from the Tools menu.

All snap tools function as follows: If no command is currently active, you can adjust the running snap tool. This means that the snap type is always active. If you are in mid-command and you select a snap tool, it becomes a "one time only" snap type. After the next click, the snap returns to the running snap. At any time you can toggle all snaps on or off by pressing F9 or double-clicking on the SNAP text in the status bar.

- Grid snap

Aligns the placement of objects to grid points. You can set the grid interval and angle in the *Draw Options* page of the *View Options* window.

- Point snap

Aligns the placement of objects to a point in the drawing.

- Midpoint snap

Aligns the placement of objects to the mid-point of another line.

- Endpoint snap

Aligns the placement of objects to the end-point of another line.

Fixture insertion points are considered as points. It is therefore possible, using the Point snap to align the placement of objects to fixtures (snap to fixtures). This is useful for dimensioning.

- Centerpoint snap

Aligns the placement of objects to the center-point of circles, arcs or cylinders.

- Assembly snap

Use assembly snap to group multiple truss objects together as you insert them. This ensures proper structural assembly. Refer to “Using Assembly snap with truss” on page 6.7-7 for more details about truss assembly.

- Pipe snap

Constrains fixtures to placement on a pipe at a specific interval. For more information, refer to “Pipe snap” on page 6.8-2.

- Focus snap

Orients one or more fixture(s) to a focus position. It also locks beam dragging to 45 degree increments of pan and 10 degree increments of tilt.

Interactive mode

Interactive mode is an alternative method for drawing objects. Objects are typically drawn using a dialog box to set the object’s size (width, depth and height for example). The full-size object is then attached to the cursor so you can place it in the drawing. There is an Interactive button in that dialog box that allows you to switch modes. Interactive mode allows you to click an insertion point for the object first, then drag to create the extents of the object as allowed by the view (XY for plan view, XZ for front view, etc.).Once those dimensions are set, a dialog box may open to allow you to set the third dimension, if required. Interactive mode works with Risers, Cylinders, Circles, Arcs, Spheres and Pipes. There are some special considerations when drawing pipes in interactive mode. This is explained in “Drawing pipes” on page 6.7-3.

To set Interactive as the default drawing mode

1. From the **Options** menu, choose **User Options**.
2. Click the *User* tab.
3. Click the *Interactive object creation* check box in the *Options* area of the window.

To set Interactive mode on the fly

Press the F11 key on your keyboard or click the Interactive tool on the *CAD Options* toolbar.

Working with DWG/DXF files

Importing DWG/DXF files

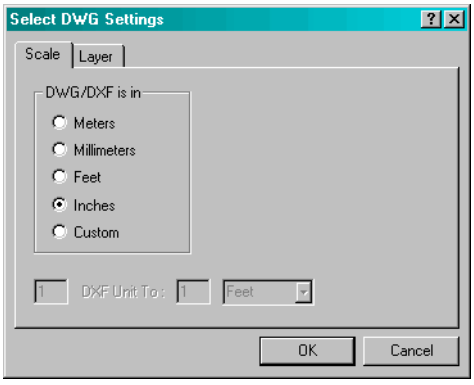
You can open or merge DWG or DXF files into your WYSIWYG show document. If you open a DWG or DXF while another show document is currently open, you will be prompted to save changes to that document before another show document is opened. Only one show document may be open at a time. Merging allows you to add the contents of the DWG or DXF file to the contents of the current document.

WYSIWYG will automatically explode the blocks in the DWG or DWG files for you if you decide to do this before opening them in WYSIWYG. Note that 3D solids are not compatible with WYSIWYG and will not import. AutoCAD files saved as release 13 or higher generally yield the best results.

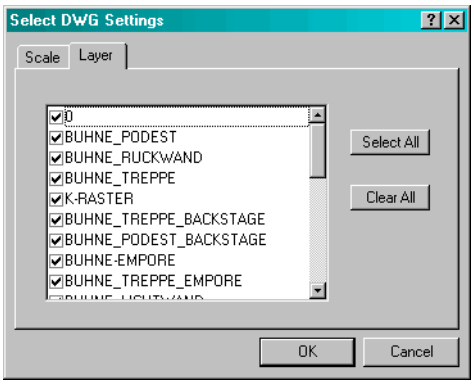
To open a DWG/DXF document

1. From the **File** menu, choose **Open**.
2. In the *File Type* box, scroll to the DWG or DXF type. This will display files of this type in the browser window.
3. Click the file name in the browser and click *Open*.

   Result: The *Select DWG Settings* dialog box is displayed.

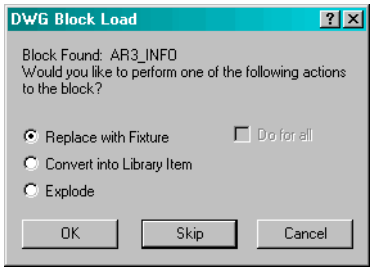


4. Select the unit type used in the DWG/DXF drawing.
5. Click the *Layer* tab and choose the layers you want to import from the DWG/DXF drawing.



6 Click *OK*.

7 If you have blocks in your drawing you have the option to explode the blocks, convert them into a library item, or substitute WYSIWYG fixtures for the existing blocks.



8 Click *OK*.

9 If you decided to substitute fixtures for blocks in step 7, you are prompted to select the fixtures from the WYSIWYG library.

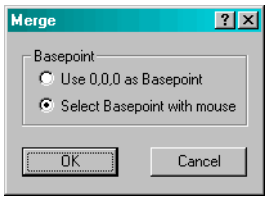
To merge a DWG/DXF document

1 From the **File** menu, choose **Merge**.

2 In the *File Type* box, scroll to the DWG or DXF type. This will display files of this type in the browser window.

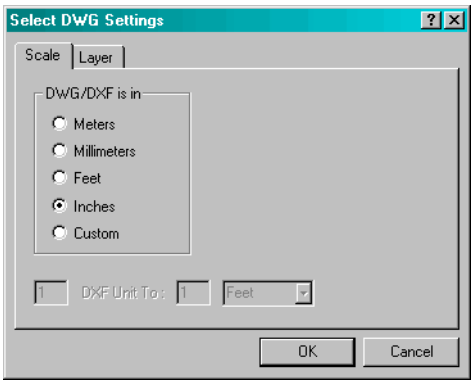
3 Click the file name in the browser and click *Open*.

Result: The *Merge* dialog box opens.



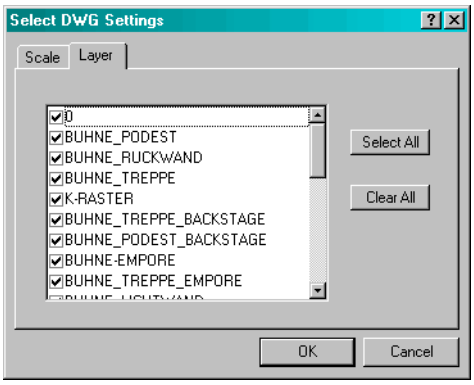
4 Choose the basepoint.

Result: The *Select DWG Settings* dialog box is displayed.



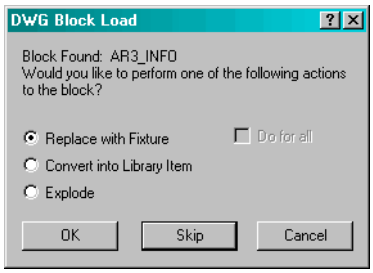
5 Select the unit type used in the DWG/DXF drawing.

6 Click the *Layer* tab and choose the layers you want to import from the DWG/DXF drawing.



7 Click *OK*.

8 If you have blocks in your drawing you have the option to explode the blocks, convert them into a library item, or substitute WYSIWYG fixtures for the existing blocks.



9 Click *OK*.

10 If you decided to substitute fixtures for blocks in step 8, you are prompted to select the fixtures from the WYSIWYG library.

11 If you chose to pick the basepoint with mouse in step 4, click to place the drawing.

Exporting DWG/DXF files

If you need to send your WYSIWYG drawings to someone using another drafting program, you can export your file to the DWG or DXF file type. DWG and DXF formats are used to transfer documents to AutoCAD or other compatible drafting applications.

To export to DWG/ DXF

1. From the **File** menu, choose **DWG/DXF Export**.
2. In the dialog that opens, use the browser to find the destination of the exported file. Ensure that the destination appears in the *Save in* box.
3. Type the name of the exported file in the *Save in* box.
4. Choose the exported file type in the *File type* box. There are a number of choices available for file type, based on Auto CAD versions.
5. Click *Save*.

   Result: The *DWG Settings dialog* box is displayed.
6. Select the type of export you want: 2D or 3D.

   Note: You cannot export a 3D drawing from WYSIWYG Report.
7. Click the *Scale* tab. Click the radio button for the units to be used in the DWG/DXF drawing.
   - **a.** To specify a custom unit of measurement, select the Custom radio button.
   - **b.** Specify the mapping of the units of measurement to use in the exported file, in the *DXF Unit To* boxes.
8. Click the *Layer* tab. Select the layers you want to include in the DWG/DXF drawing.
   - **a.** Click *Select All* to select all the layers that are listed.
   - **b.** Click *Clear All* to deselect all layers.
9. Click *OK*.

6.3 The Library Browser

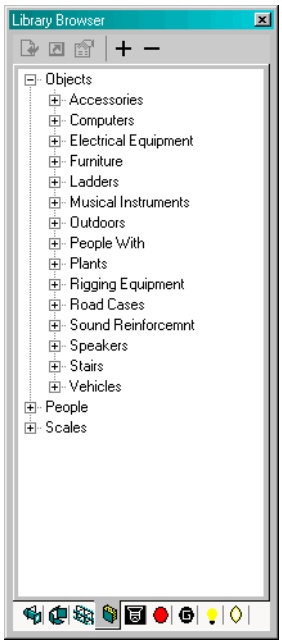
An essential component of WYSIWYG is its extensive library. You can view the contents of the library in the Library Browser.

Working with the Library Browser

To open the Library Browser

1 From the **Library** menu, choose **Browse Library**.

Tip: You can also click the *Library* tools on the *Draw* toolbar.



2 Click the tools along the bottom of the browser window to view the contents of different sections of the library. The library is divided into the following sections:

- Fixtures
- Accessories
- Truss
- Library Items (scenic)
- Symbols

- Gel (colour)
- Gobos
- Lamps
- Lenses

Library entries are divided into sort criteria. These criteria may be alphabetical, by manufacturer, by type, or by other pertinent classifications. For example, you can find the ETC Source Four in the Fixture section, Manufacturer, ETC, ERS sub-menu. You can also look in Type, Conventional, ERS, Source Four.

To view the properties of library objects

1. Open the Library Browser.
2. Navigate to the desired object.
3. Click the *Property* tool at the top of the Library Browser. .
4. Repeat steps 2-3 to view the properties of multiple objects. You may want to open the properties of different objects to compare data, such as photometric information.

Tips:

- You can also hold the ALT key and double-click on the object name.
- To view library objects without opening property windows, browse the library using the shortcut bar. To do this, select the shortcut bar for the type of object you want to view. Right-click in the shortcut bar and choose **New Shortcut**. The *Library Selection* dialog box opens with a viewing pane included.

To insert objects from the library

1. Open the Library Browser.
2. Navigate to the desired object.
3. Click the *Insert* tool at the top of the Library Browser.

   Tip: You can also double-click on the object name.

Note: Fixtures may only be placed on hang structures. Please refer to "Hanging and Focusing Fixtures" on page 6.8-1 for more information.

You can create shortcuts for frequently used objects. Shortcuts are tools found on the various shortcut bars to the left of the working area. Clicking a shortcut for an object is the same as finding the object in the Library Browser and using the *Insert* command.

To create a shortcut for a library object

1. Open the Library Browser.
2. Navigate to the desired object.
3. Click the *Create Shortcut* tool at the top of the Library Browser.

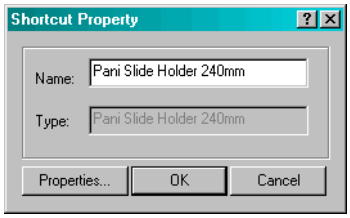
   Result: A shortcut to the object will appear on the appropriate shortcut bar.

   Tip: You can also right-click on the object name and select **Create Shortcut.**

To modify a shortcut for a library object

1. Right-click on the shortcut.
2. Select **Properties**.

   Result: The Properties dialog box is displayed. An example of the accessories *Shortcut Property* dialog box is shown below.



3. Enter a new name for the shortcut in the *Name* box.

   The name of the fixture, colour, gobo, accessories, library, or truss as listed in the library, is displayed in the *Type* box.
4. For fixtures, you can modify the type of lens and lamp associated with the fixture by selecting the appropriate option from the *Lens* and *Lamp* drop-down lists, respectively.
5. Click *Properties* to view the properties of the library entry.
6. When you are satisfied with your selections, click *OK*.

Custom library items

You can create custom library items, custom conventional fixtures, custom gobos, and custom colour and gobo wheels or scrolls.

When creating custom gobos and custom library items, you are given the option to make the item or gobo available to other documents in which case the item or gobo will be saved to the application's library. Saving to the application's library allows you to use the item or gobo in another show file. If you do not choose to make the items available to other documents, the items will only be saved within the one document. In either case, the items will be saved with the document and will be visible in the document regardless of what computer (or application) the document is opened in.

Custom conventional fixtures must be created from the WYSIWYG welcome screen and are saved directly to your application's library. The new conventional fixture will be available for you to use in all documents created using this application. If used in a show document, the custom conventional fixture will be available in the document regardless of what computer (or application) the document is opened in.

Custom colour and gobo wheels and scrolls are only saved in the document and cannot be saved to the application's library. They will travel with the document, but if you need to use the same wheel or scroll on another show, you will have to regenerate it every time.

Note: Once created, custom library items, custom gobos, and custom conventional fixtures cannot be deleted. The only way to delete them is to delete the appropriate .cdb file saved in the WYSIWYG library folder. This file is generated for each category of custom items and is not generated for each item. Deleting the gobos.cdb file, for example, deletes all of your custom gobos.

To create a custom library item

Refer to page 6.4-10.

To create a custom fixture

Refer to page 6.8-2.

To create a custom gobo

Refer to page 6.9-5.

To create custom colour or gobo lists

Refer to page 6.9-2.

6.4 Drawing Objects

In WYSIWYG, the **Draw** menu lists the objects that you can draw. Keep the following tips in mind when you are drawing the various objects.

Drawing tips

- At any time, instead of clicking points with the mouse, you can type in the desired coordinates.
- Create shortcuts for library objects.
- Use the missing coordinate.

Drawing a venue

To begin your drawing, you will want to insert a venue.

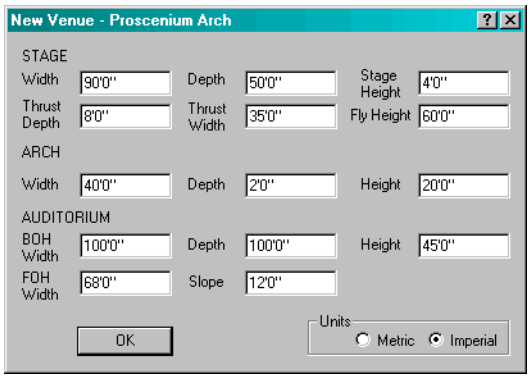
To insert a venue

1 From the **Draw** menu, choose **Venue**.

Result: A sub-menu will appear prompting you to choose the type of venue to be inserted.

1 Select **Black Box**, **Proscenium Arch**, or **Arena**.

Result: A dialog appears in which you can modify the properties of the room. The *Proscenium Arc*h dialog box is shown below.



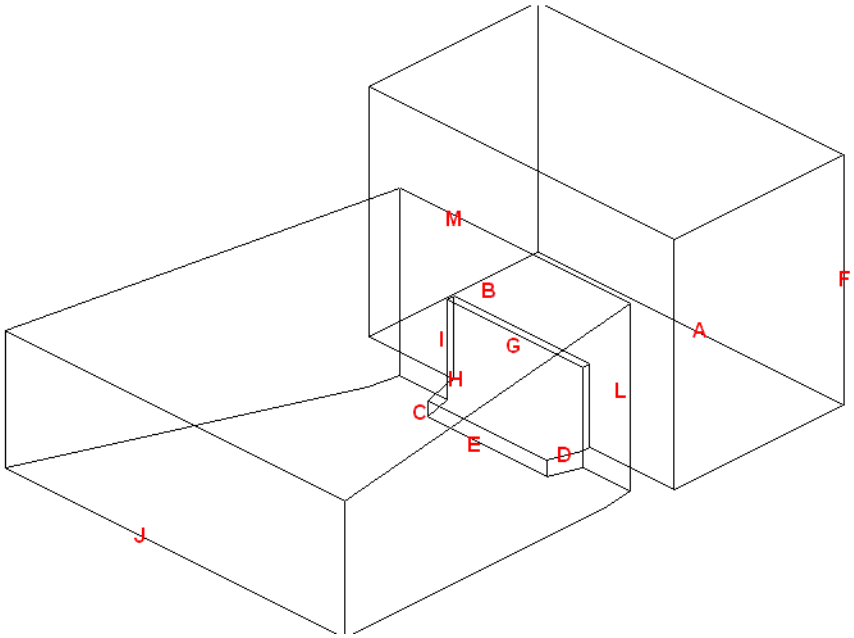
2 Click *OK*.

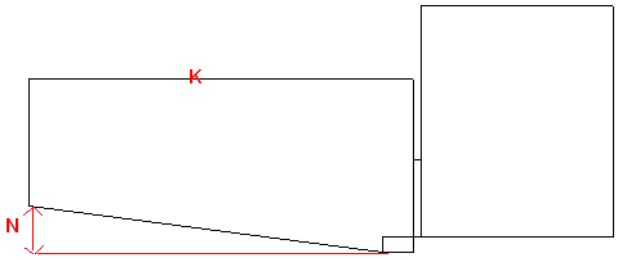
Result: The selected venue is inserted into the drawing.

The venue types are defined as follows:

Proscenium Arch

A proscenium arch venue is a basic theatrical space with a stage, proscenium arch, and an auditorium. The origin is set at the intersection of the centre line and the proscenium line on the stage floor.





The following entries describe the parts of the proscenium arch.

A: is the stage width; the dimension from the extremety of the SR wing to the extremety of the SL wing. In WYSIWYG, the default value is 90’0”.

B: is the stage depth; the dimension from the proscenium line to the back of the stage space. In WYSIWYG, the default value is 50’0”.

C: is the stage height; the dimension from the auditorium floor to the stage deck. In WYSIWYG, the default value is 4’0”.

D: is the thrust depth; the dimension from the edge of the stage to the proscenium line. In WYSIWYG, the default value is 8’0”.

E: is the thrust width; the dimension from the extremety of the SR edge of the thrust to the SL extremety of the thrust. This dimension can not be greater than the Arch width (G). In WYSIWYG, the default value is 35’0”.

F: is the Fly height; the dimension from the stage deck to the top of the fly house. In WYSIWYG, the default value is 60’0”.

G: is the proscenium arch width. In WYSIWYG, the default value is 40’0”.

H: is the depth of the proscenium arch wall. In WYSIWYG, the default value is 2’0”.

I: is the height of the proscenium arch; the dimension of the opening of the proscenium arch. In WYSIWYG, the default value is 20’0”.

J: is the back of house (BOH) width; the dimension of the auditorium from the extremety of house right to the extremety of house left at the back of the house. In WYSIWYG, the default value is 100’0”.

K: is the auditorium depth; the dimension from the proscenium line to the back of the house. In WYSIWYG, the default value is 100’0”.

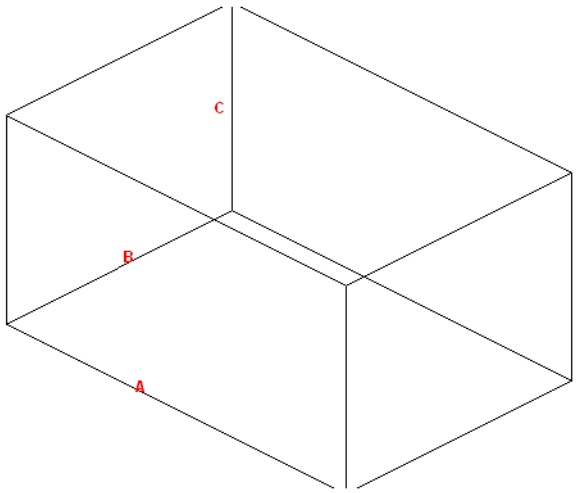
L: is the auditorium height; the dimension from the ceiling of the auditorium to the floor of the auditorium at the edge of the stage. In WYSIWYG, the default value is 45’0”.

M: is the front of house (FOH) width; the dimension of the auditorium from the extremety of house right to the extremety of house left at the front of the house. In WYSIWYG, the default value is 68’0”.

N: is the auditorium slope height. In WYSIWYG, the default value is 12’0”.

Black Box

A black box venue is an empty rectangular room. The origin is set at the centre of the room on the floor.



The following entries describe the parts of the black box.

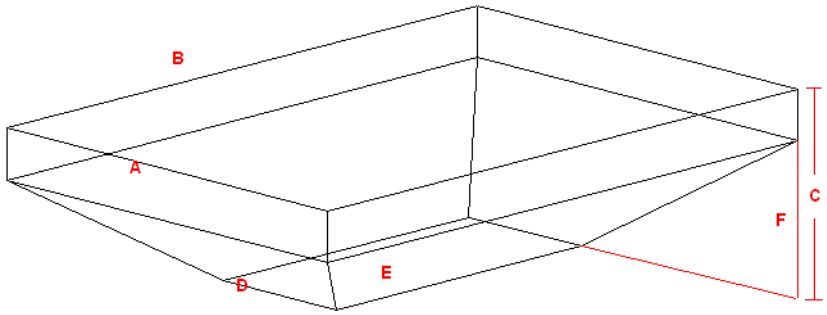
A: is the width of the room. In WYSIWYG, the default value is 60’0”.

B: is the depth of the room. In WYSIWYG, the default value is 40’0”.

C: is the height of the room. In WYSIWYG, the default value is 30’0”.

Arena

An arena venue is a basic stadium space. The following entries describe the parts of the arena. The origin is set at the centre of the venue on the floor or ice rink.



A: is the width of the arena. In WYSIWYG, the default value is 170’0”.

B: is the depth of the arena. In WYSIWYG, the default value is 250’0”.

C: is the total height of the arena. In WYSIWYG, the default value is 60’0”.

D: is the width of the floor or ice rink. In WYSIWYG, the default value is 60’0”.

E: is the depth of the floor or ice rink. In WYSIWYG, the default value is 130’0”.

F: is the height of the stands or seating; the portion of the total height that is occupied by seating or stands. In WYSIWYG, the default value is 40’0”.

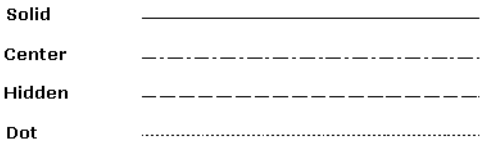
Drawing lines

Lines are 2D objects. Lines are drawn continuously, allowing you to create shapes made up of multiple vertices.

Lines come in four styles:

- Solid
- Center
- Hidden
- Dot

Line styles determine how line objects appear on your plot and can be modified at any time. They look similar to the following.



You can make a multi-vertex line appear as a spline or french curve by selecting the option in the properties box. You can then drag the markers around to adjust the curves. Alternatively, you can choose to draw a spline directly. Splines have the same line style options as regular lines.

Lines can be extruded into surfaces. For more on extruding lines, refer to “Extrude” on page 6.6-7.

To draw a line

1. From the **Draw** menu, choose **Line**.
2. From the sub-menu, select **Solid**, **Dot**, **Center**, **Hidden** or **Spline**.

   Tip: You can also use the appropriate line tool on the *Draw* toolbar. The available line tools are as follows:

   - Solid
   - Dot
   - Center
   - Hidden
   - Spline
3. Click on the drawing at the starting point of the line.
4. Drag the next vertex to its end point and click.
5. Continue to place vertices of the line as needed.
6. To end the line, right-click and choose **Finish Line** to end the line at its last end point. To abort the line entirely, choose **Abort Line**. This will erase the whole line from the drawing.

Drawing points

Points are identifiers of a specific coordinate in 3D space. Points are inserted as references or as scenic elements.

To draw a point

1. From the **Draw** menu, choose **Point**.

   Tip: You can also use the *Point* tool on the *Draw* toolbar.
2. Click on the drawing to place the point.

Points can be drawn as a dot, a cross, a square, or a circle.

To set the point type default

1. From the **Options** menu, choose **Show Options**.
2. Click the *Object Defaults* tab.
3. Select the desired point type.

Drawing circles

To draw a circle

1. From the **Draw** menu, choose **Circle**.

   Tip: You can also use the *Circle* tool on the *Draw* toolbar.
2. In the *Radius* box, enter a radius for the circle and click *OK*. The default value is 4’0”.
3. Click on the drawing to place the circle.

To draw a circle in interactive mode

1. From the **Draw** menu, choose **Circle**.

   Tip: You can also use the *Circle* tool on the *Draw* toolbar.
2. Click to place the centre of the circle.

3 Click to set the radius for the circle.

Drawing arcs

To draw an arc

1 From the **Draw** menu, choose **Arc**.

Tip: You can also use the *Arc* tool on the *Draw* toolbar.

2 In the *Radius* box, enter a radius for the arc. The default value is 4’0”.

3 In the *Start Angle* box, enter the start angle. The default value is 0.00.

4 In the *End Angle* box, enter the end angle for the arc. The default value is 180.00.

5 Click *OK*.

6 Click on the drawing to place the arc.

To draw an arc in interactive mode

1 From the **Draw** menu, choose **Arc**.

Tip: You can also use the *Arc* tool on the *Draw* toolbar.

2 Click to place the start point of the arc.

3 Click to place the end point of the arc.

4 Click to place the middle point of the arc.

Drawing text labels

To draw a text label

1 From the **Draw** menu, choose **Text Label**.

Tip: You can also use the *Text Label* tool on the *Draw* toolbar.

2 Enter the desired text in the *Text* field.

3 Enter a height for the text. This is similar to selecting a font size. The default value is 2’0”.

4 Click on the drawing to place the text label.

Note:

- You cannot change the font for text labels in CAD mode.
- The insertion point for the text label is at the intersection of the crosshairs.

To set alignment for a text label

1 Select the text label for which you want to change the justification.

2 Right click and choose **Properties**.

3 Set the horizontal and vertical justification as desired.

4 Select the *Align to View* check box to ensure the text label is legible in all view types (plan, left, right, front, back, and isometric).

5 Click *OK*.

Result: The text label will be relocated around the insertion point based on the options selected.

Drawing dimension lines

Dimension lines are 2D objects. They have a start and end point. They will measure and display the distance covered based on the measurement mode selected. The measurement modes available are: X, Y, XY, XZ, YZ, and XYZ. The dimension will only be visible in one view type. The view type is set when the dimension is drawn and is dependent on the plot type and workplane in which the dimension is drawn.

To draw a dimension

1. From the **Draw** menu, choose **Dimension.**

   Tip: You can also click the *Dimension* tool on the *Draw* toolbar.
2. Click on the drawing to set the dimension start point.
3. Right click to set the measurement mode.
4. Click on the drawing to set the dimension end point.
5. Click and drag the grab point in the center of the dimension text to drag the text to the correct side of the object, if required.

Tip:
Use snaps to connect dimensions directly to other objects.

Drawing surfaces

Surfaces are 2D objects. Surfaces are drawn continuously, stretching out behind the cursor. Surfaces can be used to create backdrops or other flat objects. Surfaces can be extruded to create odd shaped risers, or other custom shaped 3D objects. For more on extruding, refer to "Extrude" on page 6.6-7.

To draw a surface

1. From the **Draw** menu, choose **Surface**.

   Tip: You can also click the *Surface* tool on the *Draw* toolbar.
2. Click on the drawing at the starting point of the surface.
3. Drag to the next point of the surface and click.
4. Continue to place points for the surface as needed.
5. To finish and close the surface, right-click and choose **Finish Surface** from the sub-menu. To abort the surface entirely, choose **Abort Surface**. This will erase the whole surface from the drawing.

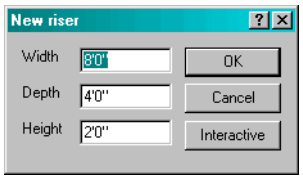
Drawing risers

Risers are solid 3D rectangular objects. Risers can be used for creating platforms, square columns, or any other box-shaped objects.

To draw a riser

1. From the **Draw** menu, choose **Riser.**

   Tip: You can also click the *Riser* tool on the *Draw* toolbar.



2 In the *Width* box, enter the width for the riser. The default value is 8'0"

3 In the *Depth* box, enter the depth for the riser. The default value is 4'0".

4 In the *Height* box, enter the height for the riser. The default value is 2'0".

5 Click *OK*.

Result: The riser will be attached to the cursor.

6 Drag the new riser to its position and click to place it in the drawing.

To draw a riser in Interactive mode

1 From the **Draw** menu, choose **Riser**.

2 Click the starting point of the riser on the drawing. The insertion point is the lower left corner of the riser.

3 Drag up and to the right to stretch out the riser's shape. Click to place the upper right corner of the riser.

4 In the dialog box that opens, enter the missing dimension of the riser.

5 Click *OK*.

Drawing cylinders

Cylinders are solid 3D cylindrical objects. Cylinders can be used for creating platforms, columns, or any other cylinder-shaped objects.

To draw a cylinder

1 From the **Draw** menu, choose **Cylinder**.

Tip: You can also click the *Cylinder* tool on the *Draw* toolbar.

2 In the *Height* box, enter the height for the cylinder. The default value is 8'0".

3 In the *Width* box, enter the width for the cylinder. The default value is 8'0".

4 Click *OK.*

Result: The cylinder will be attached to the cursor.

5 Drag the new cylinder to its position and click to place the object in the drawing.

To draw a cylinder in Interactive mode

1 From the **Draw** menu, choose **Cylinder**.

2 Click to place the centre of the bottom surface of the cylinder.

3 Click to set the radius for the cylinder.

4 In the dialog box that opens, enter the desired height or length for the cylinder.

5 Click *OK*.

Drawing spheres

Spheres are solid 3D circular or oval objects.

To draw a sphere

1 From the **Draw** menu, choose **Sphere**.

Tip: You can also click the *Sphere* tool on the *Draw* toolbar.

2 In the *Horizontal Diameter* box enter the horizontal diameter for the sphere. The default value is 8'0".

3 In the *Vertical Diameter* box enter the vertical diameter for the sphere. The default value is 8'0".

4 Click *OK*.

Result: The cylinder will be attached to the cursor.

5 Drag the new sphere to its position and click to place the object in the drawing.

Drawing pipes

For information on drawing pipes, refer to page 6.7-3.

Drawing masking

Masking, specifically borders and legs, can be automatically attached to pipes using the border and legs tool.

To draw borders and legs

1 Select the pipe(s) for which you want to attach a border and/or legs.

2 From the **Tools** menu, choose **Border and Legs**.

3 Select the *Border* check box to add a border to the selected pipe(s), and specify the height of the border.

Note: This is the distance from the pipe to the bottom of the border.

4 Select the *Legs* check box to add legs to the selected pipe(s), and specify the width and bottom elevation of the legs.

Note: The width is the distance from the extremity of the pipe to the inside edge of the leg. The bottom elevation is the distance from the floor (z = 0) to the bottom of the legs.

5 Select the *Group to pipe* check box to group the border and legs to the pipe. When this option is selected all the components (border, legs, and pipe) will be selected as one.

Note: If at a later time you want to edit a border or leg individually, you must ungroup the objects.

6 Click *OK*.

Result: Three surfaces are drawn and attached to the selected pipe.

Drawing focus positions

For more information on drawing focus positions, refer to page 6.8-19.

Drawing cameras

Cameras are set up in the drawing to provide additional view points in shaded views and renderings.

To insert a camera

1 From the **Draw** menu, choose **Camera**.

2 Enter a name for the camera and a value in degrees for the field angle. The default value for the field angle is 53.13.

3 Click on the drawing to place the camera.

4 Drag the cursor to aim the camera and click to place the target.

Result: The camera is inserted into your drawing and available in the camera selection list for shaded views and renderings.

Tip: Use the missing coordinate tool while placing the camera and target.

Note: The target of your camera is easily identifiable in Wireframe views as defined by the marker points of the camera drawing. Camera drawings are invisible in shaded views. Rather, you are looking through the camera. For this reason you can choose to display the target if you want. For more information on displaying the target, refer to "Modifying shaded views" on page 4.6-1.

Drawing a compass

A compass is added to a drawing to define the direction that is north. This definition is important to set the position of the sun and/or moon for renderings that use environmental options.

There can be only one compass in a file.

To draw a compass

1 From the **Tools** menu, choose **Define North**.

2 Click to place the starting point of the compass.

3 Click to place the end point of the compass. The end point must be directly north of the start point.

Result: A compass is drawn, centered on the start point and pointing towards the end point.

4 To remove the compass from the drawing, choose **Clear Direction** from the **Tools** menu.

5 To redefine north, repeat steps 1-3. It is unnecessary to remove the compass from the drawing when redefining as there can be only one compass in a file.

Saving custom library items

Custom library items are made of one or more objects including existing library items. Save a custom library item for a complex object that you have drawn and think you might want to use again.

Note: Fixtures, hanging structures such as pipes and truss, and splines cannot be saved as components of custom library items.

To create a custom library item

1 Select the object(s) to be saved as a custom item.

2 From the **Library** menu, choose **Create New Library Item**.

Tip: If the menu item is grayed out, one or more of the objects selected cannot be included in the custom library item.

3 Pick a base point for the item. This will become the insertion point for the item.

Result: The Library Item Wizard opens displaying a rotating shaded view of your new item.

4 Enter a name for the new library item.

5. To make this item available to other documents, check the appropriate box.
6. Click *Next*.
7. Add or modify the path.

   Note: The path is the location in the library where the item will be stored. An item assigned to the path "Custom/Tables" would be available in the custom section of the items library in a sub-category named Tables.
8. Click *Next*.
9. Enter all pertinent information including, manufacturer, weight, and other notes.
10. Click *Next*.
11. Enter your name.
12. Click *Finish*.

    Result: The custom item is placed in the item library as per the path specified in step 3.

6.5 Editing Objects

Selecting

Before you can manipulate objects, they must be selected. Selected objects are easily identifiable. A white box appears at the insertion point and the lines of the object become dotted. Fixtures that are selected are filled in in green and their beams can be set to either on or off.

- To select a single object, click on it.
- To select multiple objects:
  - Press and hold the CTRL key while clicking on the desired objects.
  - Draw a box from left to right to select all objects fully within the box.
  - Draw a box from right to left to select all objects fully within the box, as well as the objects partially contained in the box.
- To select all the objects on the current layer, press and hold the CTRL+SHIFT keys and type A or use the *Select Current Layer* tool on the *Selection* toolbar.
- To select all the objects on the screen, press and hold the CTRL and type A or use the *Select All* tool on the *Selection* toolbar.
- To select all the objects on one or more layers, from the **Edit** menu, choose **Select Special**, and then choose **Select Layers to Select**. You can also use the *Specify Layers* tool on the *Selection* toolbar. To select multiple layers, press and hold the CTRL key while selecting the layers.
- To set beam display for selected fixtures, set the *Show beams for selected fixtures* option. For more information on this option, refer to the "Show Details" on page 5.2-14.

Grouping/Ungrouping

Grouping objects together ensures that they will always be selected as one. For example, you may want to group an object and its corresponding focus position. If you relocate the object in the drawing, the focus position will automatically follow, as will all the light beams pointing at it, saving you time and work.

To group objects

1. Select the objects to be grouped.
2. From the **Edit** menu, choose **Group**.

   Tip: You may also use the *Group* tool on the *Tools* toolbar.

To ungroup objects

1. Select the grouped objects.
2. From the **Edit** menu, choose **Ungroup**.

   Tip: You may also use the *Ungroup* tool on the *Tools* toolbar.

Objects located on different layers may be grouped together but the resulting group can only exist on one layer.

To change or view the layer of a group

1. Select the object(s) for which you want to view the grouping.
2. Right click and select **Properties**.

   Tip: At any time, to access an object's properties, you can click on the *Properties* tool on the *Edit* toolbar.
3. Click the *Group* tab.

   Result: The layers for the objects in the group are listed with the layer for the group highlighted.

   Note: By default, the group assumes the current layer when initially created.
4. If you want to group the objects on a different layer, select the layer from the list.

Moving

WYSIWYG supports two types of move commands: relative and absolute.

Absolute coordinates are relative to the origin of the drawing. An absolute move is defined as moving an object from one specific point to another.

A relative move is defined as moving an object to a destination relative to its original position.

To move an object using the mouse

1. Select the object(s) to be moved.
2. Hover the cursor over the object until a gray box appears. Click and hold.
3. Drag the mouse to the new position and release to finish the move.

To move an object using absolute values

1. Select the object(s) to be moved.
2. From the **Edit** menu, choose **Move**.

   Tip: You may also use the *Move* tool on the *Tools* toolbar.
3. Click to set the base point for the move, or enter the coordinates of the base point and press ENTER.
4. Enter the new coordinate location for the base point and press ENTER.

To move an object using relative values

1. Select the object(s) to be moved.
2. From the **Edit** menu, choose **Move**.

   Tip: You may also use the *Move* tool on the *Tools* toolbar.
3. Type **@** and the distances to move the object and press ENTER. For example, to move an object 2'-0" stage right (X axis) and 3'-0" off the floor (Z axis), enter the following and then press ENTER:

@2’,0,3’

To move an object using distance and direction

1. Select the object(s) to be moved.
2. From the **Edit** menu, choose **Move**.

   Tip: You may also use the *Move* tool on the *Tools* toolbar.
3. Type **@**, the distance, the < sign and the direction (as an angle, positive or negative) to move the object and press ENTER. For example, to move an object 3’-6” to the left, enter the following and then press ENTER:

   @3’6”<-180

Rotating

Objects may be rotated around a base point. This allows you to place objects on angles in your drawing.

To rotate an object

1. Select the object(s) to be rotated.
2. From the **Edit** menu, choose **Rotate**.

   Tip: You may also use the *Rotate* tool on the *Tools* toolbar.
3. Click to set the centerpoint for the rotation, or type in the coordinates of the centerpoint and press ENTER.
4. Enter the rotation angle and press ENTER, or drag the mouse and click to enter the rotation angle. For example, to rotate an object 45 degrees, type 45 and press ENTER.

Mirroring

The Mirror command duplicates and reverses an object, inserting it the same distance from an axis line as the original.

To insert a mirrored object

1. Select the object(s) to be mirrored.
2. From the **Edit** menu, choose **Mirror.**

   Tip: You may also use the *Mirror* tool on the *Tools* toolbar.
3. Click on the drawing to enter the first axis point.
4. Click on the drawing to set the second axis point.

   Result: The mirrored object will be copied, reversed from the origin based upon the axis drawn.

Object Properties

All objects in WYSIWYG have properties. Properties are the characteristics or attributes that shape and define an object. Modifying object properties is a form of editing and is the only way to make changes to objects once they have been drawn (some exceptions exist).

To edit object(s) properties

1 Select the object(s) you want to edit.

2 Right-click on the object and select **Properties**.

Tip: At any time, to access an object's properties, you can click on the *Properties* tool on the *Edit* toolbar.

Result: The *Properties* dialog is displayed.

Note: All objects have general properties that affect the color and layer on which they are drawn. Objects also have properties that are particular to that type of object. For example, fixtures have unit numbers but do not have a radius. When an object is selected and its properties are displayed, a tab appears in the properties dialog box for that type of object. When you select multiple objects of different types, tabs will appear for each type of object selected.

General object properties

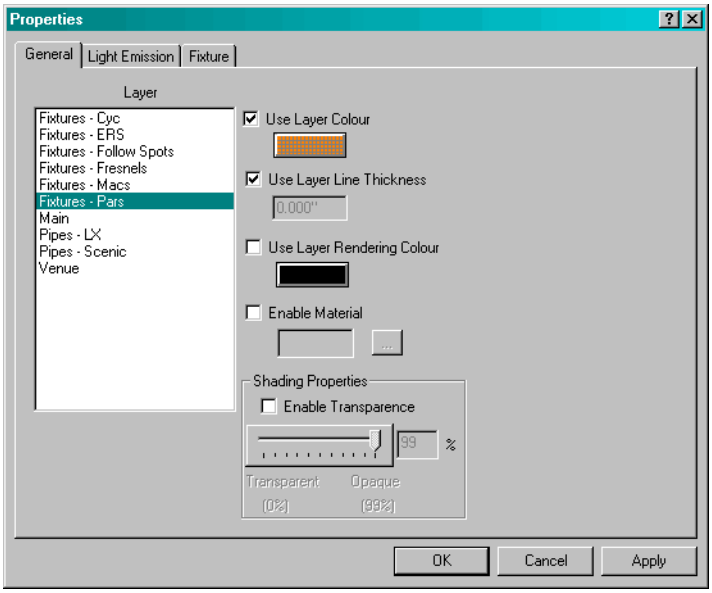
In WYSIWYG, there are two tabs that appear in the properties dialog box for every object. These two tabs are the *General* and *Light Emission* tabs.

For 3D solids and surfaces, the *Set Piece* tab also appears in the properties dialog box.

All three tabs are explained here.

1 General

Options on the *General* tab affect the selected object(s) colour, line thickness, and shading properties.



a. *Layers* list: a list of the layers in your document is displayed. Click on a list item to change the layer the object(s) will be drawn on.

b. *Use Layer Colour*: select this check box to set the object(s) colour for wireframe views to the same as the properties of the layer on which the object(s) resides. Click to clear the check box and use the colour button to set a specific colour for the object(s).

c. *Use Layer Line Thickness*: select this check box to set the object(s) line thickness to the same as the properties of the layer on which the object(s) resides. Click to clear the check box and enter a thickness in

the box below to set a specific line thickness for the object(s). The line weight is used to determine how thick the pen should be when printing a copy of the document.The applied line thickness is only visible in Print Preview mode and when printed.

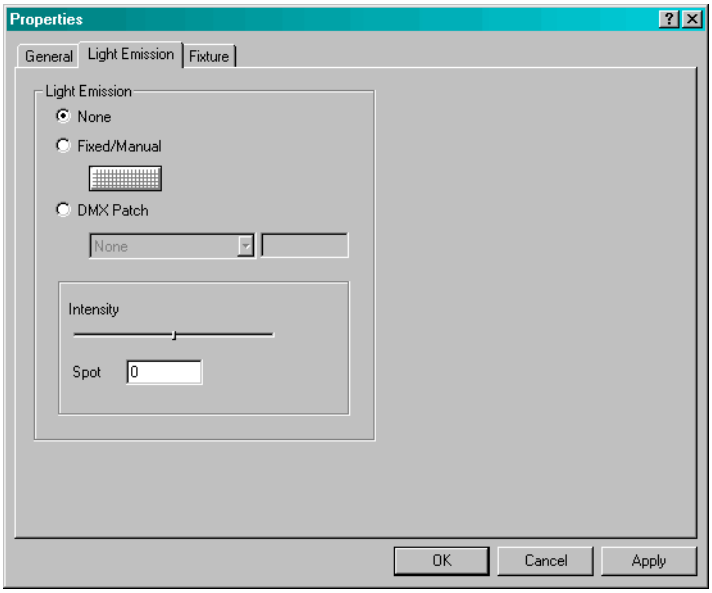
d. *Use Layer Rendering Colour*: select this check box to set the object(s) colour for shaded views and renderings to the same as the properties of the layer on which the object resides. Click to clear the check box and use the colour button to set a specific colour for the object(s).

e. *Enable Material*: select this check box to set the reflective nature of the object(s). Surfaces and objects can have a material setting applied. Click the ellipsis button (...) to select the desired material. For more information on materials, refer to page 6.5-11.

f. *Enable Transparence*: select this check box to enable the object(s) to be transparent. Use the slider to set the transparency level.

2 Light Emission

Options on the *Light Emission* tab affect the light emission properties for the selected object(s). Light emission is defined as the ability for objects to glow or emit light and can be used to simulate LEDs, lazers, neon, light boxes, projection screens, etc.



a. *None*: sets the selected object(s) to not have any light emitting properties.

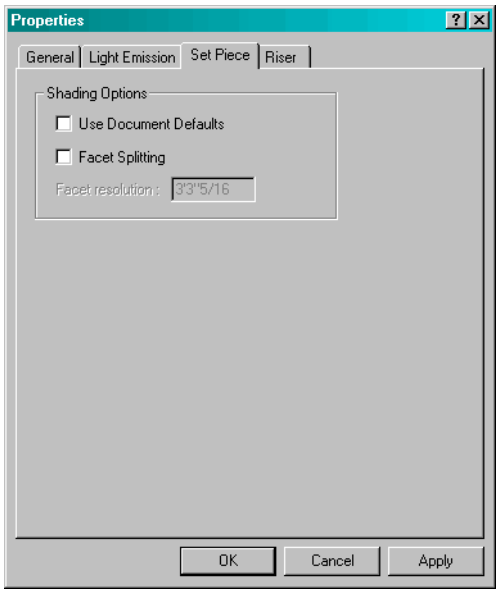
b. *Fixed/Manual*: sets the selected object(s) to have light emitting properties based on the intensity level set on the *Intensity* slider. Click the colour box to choose the colour of the light emission. Once this option is set the object will have light emitting qualities in all shaded views and renderings in all modes (CAD, Design, Live).

c. *DMX Patch*: sets the selected object(s) to have light emitting properties controllable by three DMX channels (red, blue, green). The object must be patched according to the WYSIWYG patch notation: *universe*.DMX address. Enter the patch universe of the selected object in the drop-down box. Enter the starting DMX address in the box to the right of the drop-down box. Assign a spot number for the selected object(s). This is the spot number that is used by the console to call the object. Once the object is successfully patched, you can control it from your console in Live mode as you would a regular fixture. If all channels are at a

level of 0, the object(s) is not visible. As you increase the levels on the red, blue, and green channels, the object(s) illuminates accordingly.

3 Set Piece

Options on the *Set Piece* tab affect the facet properties for the selected surface(s) and/or set piece(s) and by consequence how they will respond during simulation in shaded views when the beam quality is set to "Light up". Facet Splitting splits surface(s) and/or set piece(s) into small triangles for the purposes of smooth beam fallout.



a. *Use Document Defaults*: select this check box if you want to reference the object settings set up in *Show Options*. For more information on object settings, refer to "Object Settings" on page 5.2-8. Unselect this option to set object-specific facet options.

b. *Facet Splitting*: select this check box to ignore the document defaults and set object-specific facet properties. Specify the size of the triangles in the *Facet Resolution* box. The more triangles you have the more realistic the cosine distribution will be. However, this also decreases the simulation speed.

Object-specific properties

As explained above, objects also have properties that are particular to that type of object. For example fixtures have unit numbers but do not have a radius. When an object is selected and its properties are displayed, a tab appears in the properties dialog box for that type of object. When you select multiple objects of different types, tabs will appear for each type of object selected. The following objects are defined here:

- points
- lines
- text labels
- dimensions
- library items

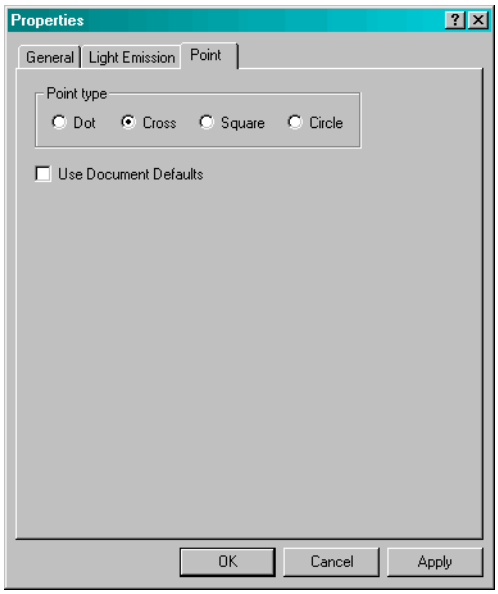
For all other types of objects (venues, circles, arcs, risers, cylinders, spheres, and cameras), please refer to the "Drawing Objects" section of this chapter starting on page 6.4-1. In these cases the properties dialog offers the same options that were given when the object was initially drawn.

Hanging structures properties (pipes, truss, floor mounts, etc.) are fully defined in the "Hang Structures" section of this chapter starting on page 6.7-1.

Fixtures and lighting-specific object properties are fully defined in the "Hanging and Focusing Fixtures" section of this chapter starting on page 6.8-1.

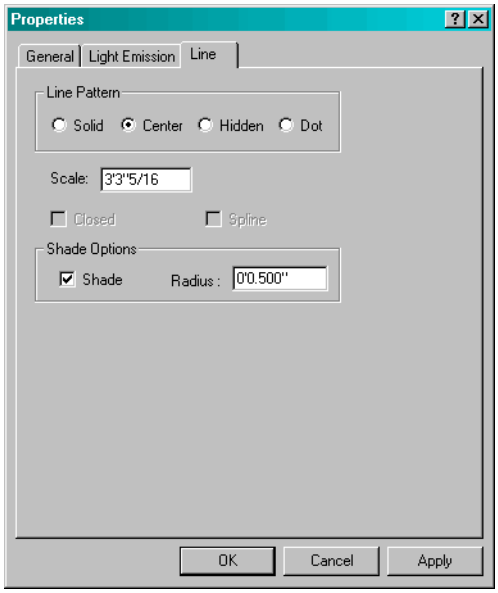
1 Points

Options on the *Point* tab affect how the selected point(s) is drawn. The default point type used in a document is defined on the *Object Defaults* tab of *Show Options*. To ignore the default setting click to clear the *Use Document Defaults* check box and select the desired point type.



2 Lines

Options on the *Line* tab affect how the selected line(s) is drawn.



a. *Line Pattern*: choose a line pattern for the selected line(s). Refer to "Drawing lines" on page 6.4-4 for an illustration of each type.

b. *Scale*: enter a value in the scale box to change the length and spacing of dots and dashes for the selected line(s). This is applicable to center, hidden, or dot lines only.

c. *Closed*: select the *Closed* check box to quickly connect the first point of a multi-segment line to the last point of that line.

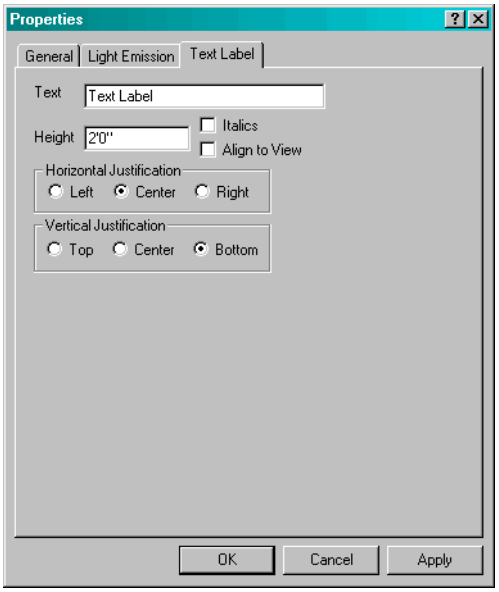
d. *Spline*: select the *Spline* check box to transform a line into a spline or french curve.

Note: You cannot change a line to a spline or french curve unless the selected line has more than two vertices.

e. *Shade Options*: select the *Shade* check box to have the selected line(s) displayed in shaded views and renderings. Enter a value for the radius to set its thickness.

3 Text labels

Options on the *Text Label* tab affect the justification and style of the selected text label(s).



a. *Text*: enter new text to change the label.

b. *Font*: enter a value for the height to adjust the font size. Select the *Italics* check box for italics.

c. *Alignment*: select the *Align to View* check box to set the text label(s) to be legible regardless of plot type.

d. *Horizontal justification*: select the desired check box to set the text label(s) horizontal alignment. The available horizontal alignment options are:

Left: Places the selected text label to the left of the insertion point.

Center: Horizontally centers the selected text label on the insertion point.

Right: Places the selected text label to the right of the insertion point.

e. *Vertical justification*: select the desired check box to set the text label(s) vertical alignment. The available vertical alignment options are:

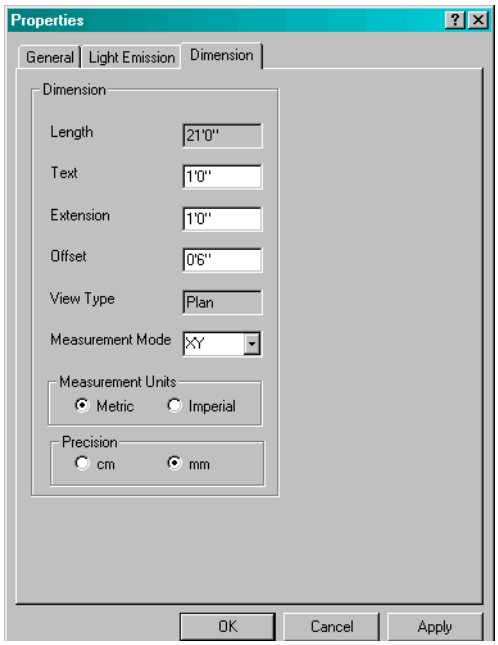
Top: Places the selected text label below the insertion point.

Center: Vertically centers the selected text label on the insertion point.

Bottom: Places the selected text label above the insertion point.

4 Dimensions

Options on the *Dimension* tab affect the appearance and measurement mode of the selected dimension(s). To change the length of a dimension, it must be stretched in the drawing.



a. *Length*: this field is not editable. It displays the actual length of the line drawn for the dimension.

b. *Text*:enter a height value to change the font size of the dimension label.

c. *Extension*: enter a value to change the length of the extension lines for the dimension. The extension lines are the lines that mark the extremeties of the dimension.

d. *Offset*: enter a value to change the offset. The offset is the distance between the extension lines and the object being measured.

e. *View Type*: this field is not editable. It displays the plot type in which the dimension label is visible which is determined when the dimension is drawn.

f. *Measurement Mode*: select an option to change the measurement mode. Measurement modes are explained on page 6.4-7.

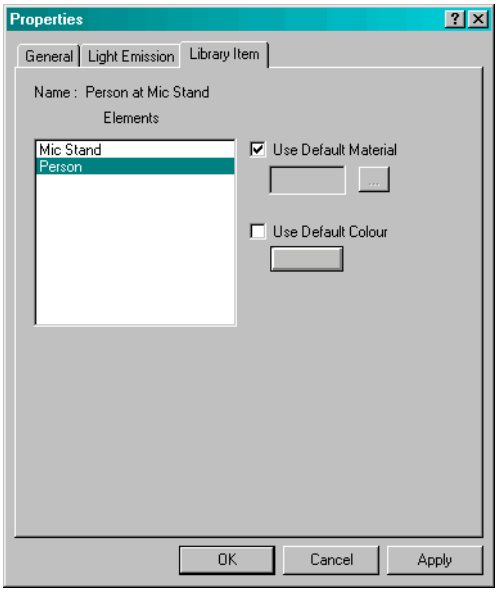
g. *Measurement Units*: select the *Imperial* or *Metric* radio buttons to change the measurement units.

h. *Precision*: select the appropriate radio buttons to specify measurement precision of the dimension line. Based on your choice for measurement units, you can choose to display the drawing resolution in centimetres, millimetres, whole numbers, or fractions.

5 Library items

A library item is comprised of one or more elements. Options on the *Library Item* tab allow you to change the colour and material for those elements.

Library items can have different materials and colours applied to each element. For example, the leaves and planter of the Potted Plant object can have different materials applied to them.



To specify or change the material and/or colour used for each element, highlight the element in the list, click to clear the *Use Default* box and click the ellipsis (...) button to choose a new material or colour button to select a colour. For more information on materials, refer to page 6.5-11.

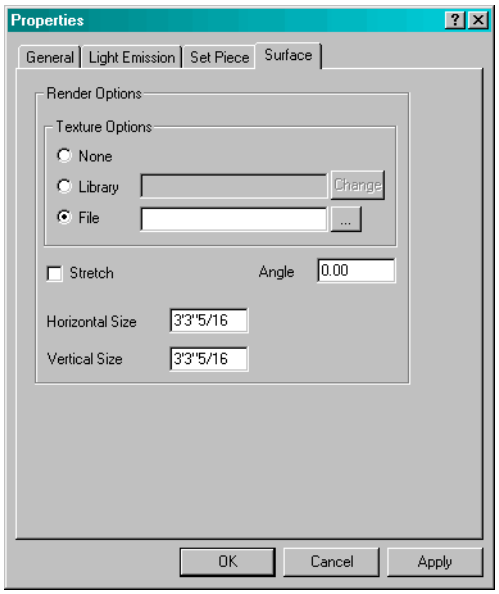
Textures

You can apply textures to surfaces in the *Properties* dialog box. Textures are provided with WYSIWYG that emulate various types of surfaces, like brick or cloth. You can also use your own bitmap or jpeg files as textures, to emulate projection screens, or to cover a surface with a texture specific to your show.

Note: Availability of textures depends on the type of installation performed. Textures are installed in a "Full" installation from the CD. If you performed a "Typical" installation, the textures will not be immediately available. Run the installer again and choose *Modify* to load the texture files. If you downloaded WYSIWYG from the CAST Web site, you will need to download and install the textures file separately.

To apply a texture to a surface

1. Right-click the surface you want to texture.
2. Choose **Properties**.
3. Click the *Surface* tab.



4 Select the texture option that you want. The options are:

- *None*. Indicates that no texture has been applied to the selected surface(s) or removes any texture from the surface.
- *Library*. Indicates that the selected surface is a texture from the WYSIWYG library. Click the Change button to modify the selected surface.
- *File*. The selected surface will use a texture from an external bitmap or jpeg file on your hard drive rather than one from the WYSIWYG library. Click the ellipsis (...) button to select the external file to be used as a texture.

5 Select the *Stretch* check box if you want the texture selected above to grow to cover the entire surface. Based on the aspect ratio and the rotation angle, WYSIWYG will evaluate the surface with all of its edges and stretch the texture so the best fit is used. This check box is not active unless a texture is chosen above.

6 If the texture has text or another recognizable image in it, you may need to rotate the image to get it right-side up. Enter the rotation angle in the *Angle* box.

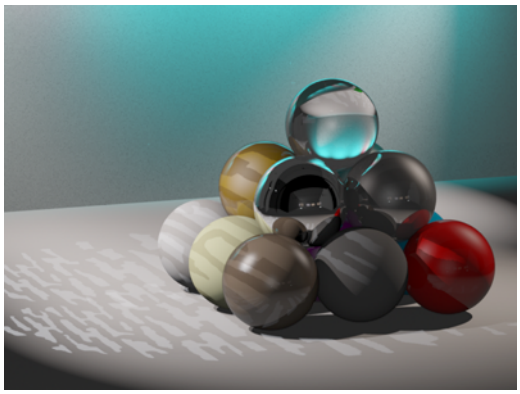
7 If Stretch is not used, the *Horizontal Size* box allows you to set the real world size of the horizontal distance the selected texture will cover on the selected surfaces before the texture tiles (repeats itself). This check box is not active unless a texture is chosen above.

8 If Stretch is not used, the *Vertical Size* box allows you to set the real world size of the vertical distance the selected texture will cover on the selected surfaces before the texture tiles (repeats itself). This check box is not active unless a texture is chosen above.

9 Click *OK*.

Materials

Materials differ from textures in that they do not affect what the objects or surfaces will look like but rather how they will interact with light. The result of applying a material is only apparent in renderings, as shown below.

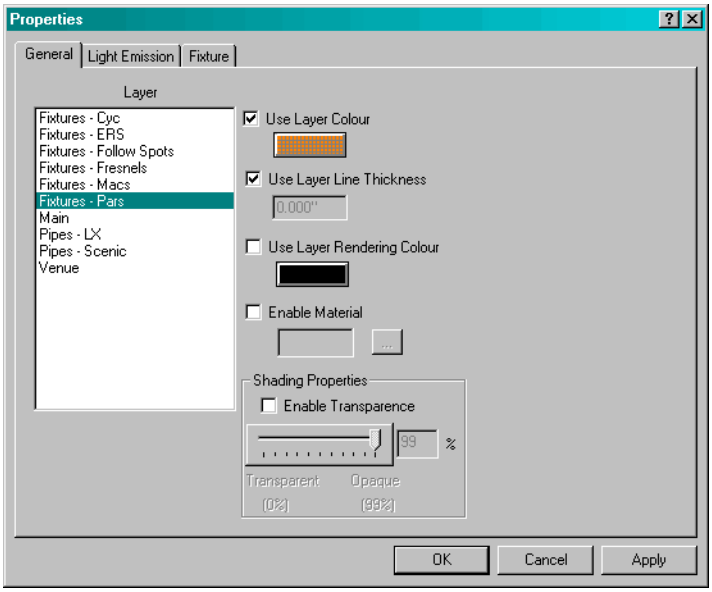


Objects and surfaces can be assigned both a texture and a material.

You can apply materials to surfaces and objects using the *General* tab in the *Properties* dialog box. A list of materials is provided in WYSIWYG from which you can choose the desired material.

To apply a material to an object or surface

1. Right-click the surface or object to which you want to apply material.
2. Choose **Properties**.
3. Click the *General* tab.



4. Select the *Enable Material* check box.
5. To choose the material, click the ellipsis (...) button.
6. Select the material and then click *Select*.
7. Click *OK*.

Applying a material to a Library item element

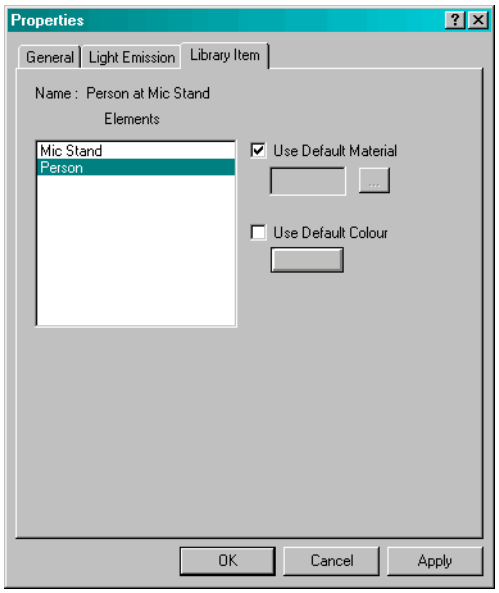
To apply a material to a Library item element

Library objects can further have different materials applied to each element of the object using the *Library Item* tab in the *Properties* dialog box.

1 Right-click the library item to which you want to apply material.

2 Choose **Properties**.

3 Click the *Library Item* tab.



4 Select an element from the *Element* list.

5 If a material was chosen previously (on the *General* tab), the *Use Default Material* check box is selected. To choose a different material, click to clear the *Use Default Material* check box. and then click the ellipsis (...) button to choose another material.

6 Repeat steps 4 and 5 for each element to which you want to apply materials.

7 When finished, click *OK*.

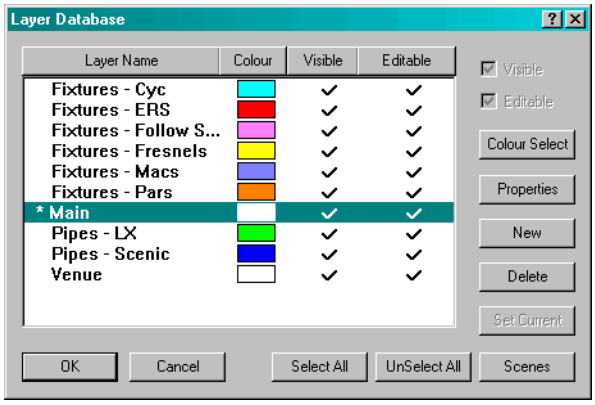
Layers

Layers are drawing aids intended to help you organize your plot. One way to think of layers is as transparent acetate sheets upon which you can draw. In the same way that you can view several transparent sheets at once by placing them on top of each other you can hide and unhide layers by choosing which sheets are in the stack. The top sheet is the layer that new objects will be placed on and is called the current layer.

To create new layers

1 From the **Edit** menu, choose **Layers**.

Result: The *Layer Database* dialog box is displayed.



2 To create a new layer click *New*.

3 Enter a new name for your layer and then click *OK*. For example, "Pipes".

4 To change the colour of the layer, click *Colour Select*. It is recommended that you select different colours for each layer so as to easily identify the layers on the plot.

5 To make a layer and all objects on the layer visible on the plot and in shaded views, select the *Visible* check box. If not selected, the layer is not visible and therefore not editable.

6 To make a layer and all objects on the layer editable on the plot and in shaded views, select the *Editable* check box. If not selected, the layer is not editable and therefore not visible.

Tip: You might want to deselect a layer when you are finished working with it to avoid possible errors while working on other parts of the plot. This is similar to freezing in Auto CAD.

7 To view and modify the scenes a layer is included in, select a layer and then click *Scenes*. Scenes are described on page 6.5-15.

8 To view the properties of a layer (for example, name, colour, and inventory of objects drawn on the layer), click *Properties*.

9 Before closing the dialog, set your current layer by highlighting it and then clicking *Set Current*.

The current layer is the layer that you are working on at the moment – any object that you draw will be placed on this layer and assume the layer's default properties when it is drawn. An asterisk (*) appears beside the name of the current layer.

10 Click *OK* to close the Layers dialog.

Note: To quickly sort the layers within the layer database, click the appropriate column heading in the *Layer Database* dialog box. For example, click the *Editable* column heading to sort the layers by edit setting; that is, those layers marked as editable appear at the top of the list in alphabetical order.

To delete a layer

You must remove all objects from a layer before you can delete it. You cannot delete the current layer.

1 Select all of the objects on the layer that you want to delete and delete them or move them to another layer.

2 From the **Edit** menu, choose **Layers**.

Result: The *Layer Database* dialog box is displayed.

3 Ensure that the layer you want to delete is not set as the current layer.

4 Select the layer in the list.

5 Click *Delete*.

6 Click *OK*.

Tips:

- There is a toolbar named *Layer* that gives you quick access to all existing layers and the Layers database. Click the tool on the left end of the *Layer* toolbar to access the Layer database.
- Use the layer list in the *Layer* toolbar to quickly set the current layer. If objects are selected when you do this, those objects will be moved to the selected current layer.

Scenes

Scenes are collections of layers used to organize the plot. A scene may contain one or more layers.

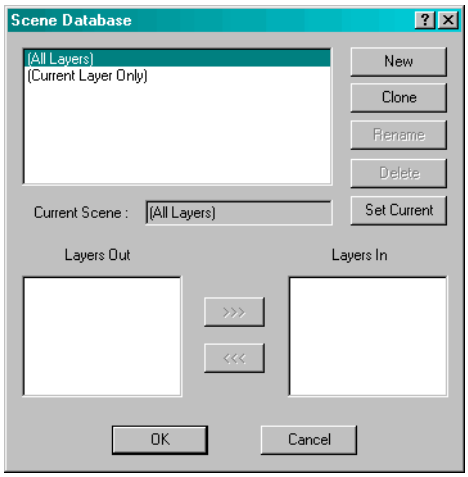
Hiding a scene hides all the layers assigned to that scene. By organizing the layers and scenes in your plot you can quickly perform set changes.

By default WYSIWYG creates two initial scenes: *Current Layer Only* and *All Layers*. You cannot delete these scenes.

To create a new scene

1 From the **Edit** menu, choose **Scenes**.

Result: The *Scene Database* dialog box is displayed.



2 Click *New* to create a new scene.

3 Enter the name of the new scene and then click *OK*.

Result: The layers associated with the scene are displayed in the *Layers Out* and *Layers In* boxes.

4 Select the layers you want to include in the scene and use the arrow buttons to move the layers accordingly. The layers that are available in a scene can also be set in the *Layers* dialog box. For more information, refer to "Layers" on page 6.5-13.

5 If you want to copy a scene, select the scene and click *Clone*. The copied scene has the same properties of the selected scene.

6 To rename a scene, select the scene and click *Rename*.

7 To remove a scene from the scene database, select the scene and click *Delete*.

8 Click *Set Current* to activate your new scene. If the current layer is not in the current scene, you will not be able to add any objects.

9 Click *OK*.

Tip:
There is a toolbar named *Scene* that gives you quick access to all existing scenes and the Scenes database. Click the tool on the left end of the *Scene* toolbar to access the Scene database.

Note: In order for fixtures to be displayed in scenes, both the layer on which the fixtures reside and the layer on which the respective hang structure resides, if different, must be included in the scene. You can not display fixtures without their hang structures.

6.6 CAD Tools

Break

The Break command allows you to break an object into smaller components.

To break an arc into arcs

1. Select the arc you want to break.
2. From the **Tools** menu, choose **Break**, then choose **Arc into arcs**.
3. Enter the number of desired break arcs and click *OK*.

   Result: The arc is broken equally into the number of arcs specified.

To break an arc into lines

1. Select the arc you want to break.
2. From the **Tools** menu, choose **Break**, then choose **Arc into lines**.
3. Enter the number of desired break lines and click *OK*.

   Result: The arc is broken equally into the number of lines specified.

To break a circle into arcs

1. Select the circle you want to break.
2. From the **Tools** menu, choose **Break**, then choose **Circles into arcs**.
3. Enter the number of desired break arcs and click *OK*.

   Result: The circle is broken equally into the number of arcs specified.

To break a circle into lines

1. Select the circle you want to break.
2. From the **Tools** menu, choose **Break**, then choose **Circles into lines**.
3. Enter the number of desired break lines and click *OK*.

   Result: The circle is broken equally into the number of lines specified.

To break a line at vertices

1. Select the multi-segment line you want to break.
2. From the **Tools** menu, choose **Break**, then choose **Lines at Vertices**.

   Result: The continuous multi-segment line is broken at the segment intersections. Each vertex is now an independent line.

To break a line into lines

1. Select the line or multi-segment line you want to break.
2. From the **Tools** menu, choose **Break**, then choose **Lines into lines**.
3. Enter the number of desired break lines and click *OK*.

   Result: The line is broken equally into the number of lines specified.

To break a set piece into faces

1 Select the 3D solid you want to break, a riser for example.

2 From the **Tools** menu, choose **Break**, then choose **Set pieces into faces**.

Result: Each face of the solid becomes an independent surface.

To break a set piece into lines

1 Select the 3D solid you want to break, a riser for example.

2 From the **Tools** menu, choose **Break**, then choose **Set pieces into lines**.

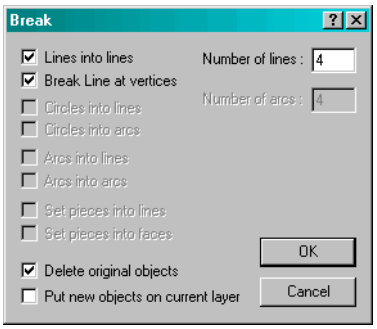
Result: Each segment of the solid becomes an independent line. The 3D solid in no longer a solid.

Alternatively, to perform a break command, you can use the *Break* tool on the *Tools* toolbar. This is the same as choosing **Specify** from the **Break** sub-menu.

To break using the Specify dialog

1 Select the object you want to break

2 From the **Tools** menu, choose **Break**, then choose **Specify**.



3 Select the desired break type from the available options depending on what object you have selected to break by selecting the appropriate check boxes.

4 In the boxes at the top right corner, specify the number of break lines or arcs if appropriate.

5 Select the *Delete original objects* check box if you want the original object to be replaced with the new "broken" ones.

6 Select the *Put new objects on current layer* check box if you want the new "broken" objects to be placed on the current layer, as opposed to the selected object's original layer if different.

7 Click *OK*.

Convert

The Convert command turns an object into a different kind of object. For example, you can convert a circle into a circular surface, you can convert lines into pipes, and vice versa, and you can convert points into focus positions, and vice versa.

To convert a circle into an arc

1. Select the circle you want to convert.
2. From the **Tools** menu, choose **Convert**, then choose **Circles into arcs**.

   Result: The circle is converted into a 360 degree arc. The marker points can be dragged out to open the arc if desired.

To convert a circle into a surface

1. Select the circle you want to convert.
2. From the **Tools** menu, choose **Convert**, then choose **Circles into surfaces**.
3. Enter the number of segments to make up the contour of the surface. The number entered cannot be 1. The higher the number the more circular the surface will be.

   Result: The circle is converted into a surface.

   Note: If the number of segments entered is 2, the circle will be converted to a linear surface. If the number of segments entered is 3, the circle will be converted to a triangle surface. If the number of segments entered is 4, the circle will be converted to a square surface. This pattern will repeat for each increment in the number of segments.

To convert a line into a pipe

1. Select the line or lines you want to convert.
2. From the **Tools** menu, choose **Convert**, then choose **Lines into pipes**.

   Result: The line(s) is converted into a pipe. It assumes all of the properties and characteristics of a pipe. It can be named and fixtures may be hung on it.

To convert a pipe into a line

1. Select the pipe(s) you want to convert.

   Note: No fixtures may be hung on the selected pipe. You must unhang or delete any fixtures before attempting to convert the pipe into a line.
2. From the **Tools** menu, choose **Convert**, then choose **Pipes into lines**.

   Result: The pipe(s) will be converted into a line.

To convert a focus position into a point

1. Select the focus position(s) you want to convert.
2. From the **Tools** menu, choose **Convert**, then choose **Focus positions into points**.

   Result: The focus position(s) will be converted into a point. Any fixtures that were focused to the initial focus position will retain their orientation but will loose the association with that point.

To convert a point into a focus position

1 Select the point(s) you want to convert.

2 From the **Tools** menu, choose **Convert**, then choose **Points into focus positions**.

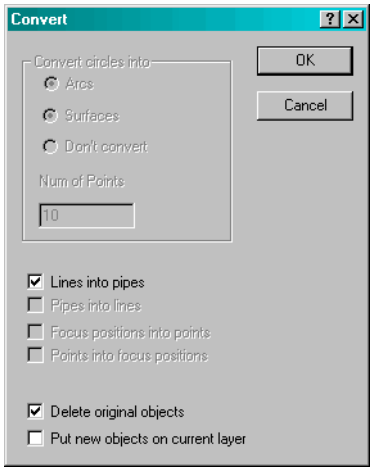
Result: The point(s) will be converted into a focus position. It will automatically be given a label entitled "Label". To change the name for the newly created focus position, refer to "Focus position properties" on page 6.8-5.

Alternatively, to perform a convert command, you can use the *Convert* tool on the *Tools* toolbar.

To convert using the Specify dialog

1 Select the object you want to convert.

2 From the **Tools** menu, choose **Convert**, then choose **Specify**.



3 Select the desired convert type from the available options by selecting the appropriate radio buttons. Select the *Don't convert* radio button if you do not want to convert the circle into either an arc or a surface.

Note: For circles select the A*rcs* or S*urfaces* radio button and enter the number of points as discussed above.

4 Select the *Delete original objects* check box if you want the original object to be replaced with the new "converted" one.

5 Select the *Put new objects on current layer* check box if you want the new "converted" objects to be placed on the current layer, as opposed to the selected object's original layer if different.

6 Click *OK*.

Divide

The divide command separates objects into equal parts. The initial object is left intact (if not deleted) and the parts are identified by newly created objects such as points or lines.

To divide an arc into lines

1. Select the arc you want to divide.
2. From the **Tools** menu, choose **Divide**, then choose **Arcs into lines**.
3. Enter the number of lines desired.

Result: The arc is divided into equal parts by the number of lines specified. The lines are automatically drawn from the center of the arc outward. What you get is similar to a pie chart.

To divide an arc into points

1. Select the arc you want to divide.
2. From the **Tools** menu, choose **Divide**, then choose **Arcs into points**.
3. Enter the number of points desired.

Result: The arc is divided into equal parts by the number of points specified. The points are placed along the arc at the calculated interval.

To divide a circle into lines

1. Select the circle you want to divide.
2. From the **Tools** menu, choose **Divide**, then choose **Circles into lines**.
3. Enter the number of lines desired.

Result: The circle is divided into equal parts by the number of lines specified. The lines are automatically drawn from the center of the circle outward. What you get is similar to a pie chart.

To divide a circle into points

1. Select the circle you want to divide.
2. From the **Tools** menu, choose **Divide**, then choose **Circles into points**.
3. Enter the number of points desired.

Result: The circle is divided into equal parts by the number of points specified. The points are placed along the circle at the calculated interval.

To divide a cylinder into circles

1. Select the cylinder you want to divide.
2. From the **Tools** menu, choose **Divide**, then choose **Cylinders into circles**.
3. Enter the number of circles desired.

Result: The cylinder is divided into equal parts by the number of circles specified. The circles are stacked within the cylinder at the calculated interval.

To divide a cylinder into lines

1. Select the cylinder you want to divide.
2. From the **Tools** menu, choose **Divide**, then choose **Cylinder into lines**.
3. Enter the number of lines desired.

Result: The cylinder is divided into equal parts by the number of lines specified. The lines follow the same pattern as the cylinder's existing segments but at the calculated interval.

To divide a line into points

1 Select the line you want to divide.

2 From the **Tools** menu, choose **Divide**, then choose **Lines into points**.

3 Enter the number of points desired.

Result: The line is divided into equal parts by the number of points specified. The points are placed along the line at the calculated interval.

Lines may also be divided by adding vertices. In this case no new objects are created; the line is simply divided by marker points.

To add a vertex to a line

1 Select the line to be divided.

2 Right-click on the line segment you want to divide and choose **Add Vertex**.

Result: The original line will remain intact. A marker point will be placed at the midpoint of the line.

Note: This procedure can be repeated for vertices within a line. The marker points will always be placed at the mid point of the vertices.

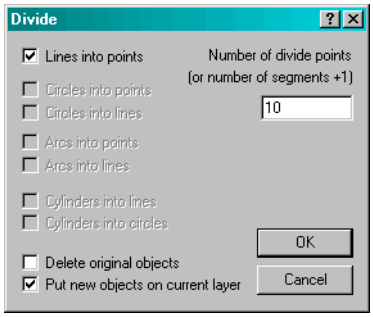
- Marker points can be dragged out to re-shape the line.
- If the line is deleted, the marker points are also deleted.

Alternatively, to perform a divide command, you can use the *Divide* tool on the *Tools* toolbar. This is the same as choosing *Specify* from the *Divide* sub-menu.

To divide using the Specify dialog

1 Select the object you want to divide.

2 From the **Tools** menu, choose **Divide**, then choose **Specify**.



3 Select the desired divide type from the available options by selecting the appropriate check boxes.

4 Select the *Delete original objects* check box if you want the original object to be deleted when the new ones are created.

5 Select the *Put new objects on current layer* check box if you want the new objects to be placed on the current layer, as opposed to the selected object's original layer if different.

6 Click *OK*.

Extrude

The Extrude command allows you to turn a 2D object like a line or an arc into a surface, or turn a surface into an extruded surface or 3D solid.

To extrude a line

1. Select the line you want to extrude.
2. From the **Tools** menu, choose **Extrude**, then choose **Lines into Surfaces**.
3. Enter the distance to extrude the line. The extrusion will occur parallel to the axis of the missing coordinate. For example, a line shown in plan view will extrude vertically. For splines, you must specify the number of segments for the new surface in the *# of Points* box.

To extrude an arc

1. Select the arc you want to extrude.
2. From the **Tools** menu, choose **Extrude**, then choose **Arcs into Surfaces**.
3. Enter the distance to extrude the line. The extrusion will occur parallel to the axis of the missing coordinate. For example, an arc shown in plan view will extrude vertically.
4. Enter the number of segments for the new surface. The number entered cannot be 1. The higher the number the more circular the surface will be.

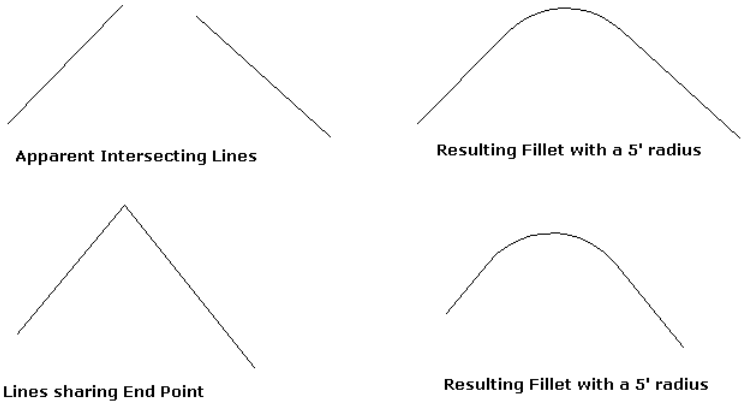
To extrude a surface into an extruded surface

1. Select the surface you want to extrude.
2. From the **Tools** menu, choose **Extrude**, then choose **Surface into Extruded Surface**.
3. Enter the distance to extrude the surface. The extrusion will occur parallel to the axis of the missing coordinate. For example, a surface shown in plan view will extrude vertically.

Tip:
You may also use the *Extrude* tool on the *Tools* toolbar.

Fillet

The Fillet command joins two lines sharing the same end point or two apparent intersecting lines with an arc. A fillet radius must be specified to achieve the desired fillet angle.



To set the fillet radius

1. From the **Tools** menu, choose **Fillet**, then choose **Specify fillet radius**.
2. Enter a radius measurement for the resulting fillet.

   Note: All subsequent fillet commands will follow this value until it is modified again.

To fillet two lines

1. Select the lines to be filleted.
2. From the **Tools** menu, choose **Fillet**, then choose **Fillet**.

   Result: The lines are joined by an arc based on the value set for the fillet radius.

Notes:

1. A fillet radius must be set to perform this command.
2. If the fillet radius is 0, the lines will be extended and joined at their intersection. If they are already intersecting, nothing will occur.

Array

The Array command is used to make multiple copies of an object distributed evenly along a line or arc.

To perform a linear array

1. Select the object you want to array.
2. From the **Tools** menu, choose **Array**, then choose **Array Linear**.
3. Click a point on your drawing or type in coordinates to specify the direction and interval for the linear array. For example, to place a copy of the selected object every 5’ at a 45 degree angle enter the following and then press ENTER:

   @5’<45

4. In the dialog box that opens, enter the total number of copies desired (this will include the original object). Adjust the value for the interval if required.

5 Click *OK*.

Note: The interval is defined as the distance between the copied objects.

To perform a polar array

1 Select the object you want to array.

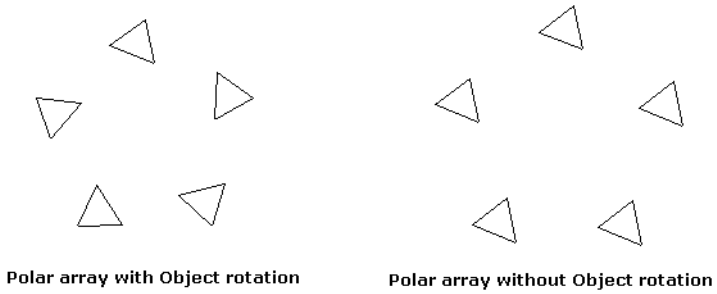
2 From the **Tools** menu, choose **Array**, then choose **Array Polar**.

3 Click a point that will represent the centre of the circle around which the objects will be arrayed.

4 In the dialog box that opens, enter the total number of copies desired (this will include the original object).

5 Specify the total angle in degrees to fill with copied objects.

6 Select the *Rotate Objects* check box if you want the copied objects to rotate as they are pasted around the center of the circular array, as shown below.



Polar array with Object rotation

Polar array without Object rotation

7 Click *OK*.

Distance

The distance tools allows you to measure the distance and angle from one point to another.

To measure a distance

1 From the **Tools** menu, choose **Distance**.

Tip: You can also use the *Distance* tool on the *Tools* toolbar.

2 Pick the starting point for the measurement.

3 Pick the end point of the measurement.

Result: The length and angle of the specified trajectory is displayed in the bottom left corner of the status bar.

Tip: When selecting the starting and end points for the measurement use the snap tools for precision.

6.7 Hang Structures

Hang structures are a very important part of creating a light plot in WYSIWYG. All fixtures must be placed on a hang structure - either a pipe or a truss. This includes fixtures on set mounts or floor mounts. Hang structures are not only a place to hang fixtures, but also a means to organize your paperwork. Some general facts about hang structures:

- Hang structures can be pipes or trusses. Pipes are drawn, trusses are placed from the truss library. There are many different types of truss pieces in the truss library, including box, triangle, and curved. Truss includes floor mount, boom, and ladder positions as well.
- AutoUnit will assign and adjust unit numbers for fixtures on a pipe as you add, move or delete them.
- Hang structures can have the same name, even if they are not connected. For example, you can use this for set mounts, where you have many small pipes attached to the same piece of scenery.

Position Manager

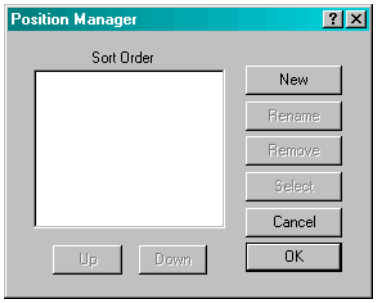
The Position Manager organizes all the available position names in your show document. It also sets the sort order of your positions. You can use the Position Manager to change the sort order of the position names, which is important for printed paperwork like instrument schedules and other reports that use the Position field. The order of position names is also used for general sorting in the Data view and the order in which positions appear in the Flight Case.

The Position Manager is accessible from the *Properties* window of a hang structure and from the **Positions** command in the **Edit** menu.

To view and edit entries in the Position Manager

1 From the **Edit** menu, choose **Positions**.

Note: The ellipsis button [...] on the Hang Structure page of a pipe's or truss' properties will also open the Position Manager.



2 Click *New*.

3 Type the name of the new position and click *OK*.

4 To modify the name of a position, select the position and then click *Rename*. Type the new name and click *OK*.

5 To remove a position name, select the name and then click *Remove*. Removing the position name does not also delete the hang structure. Rather the hang structure removes its association to the position name.

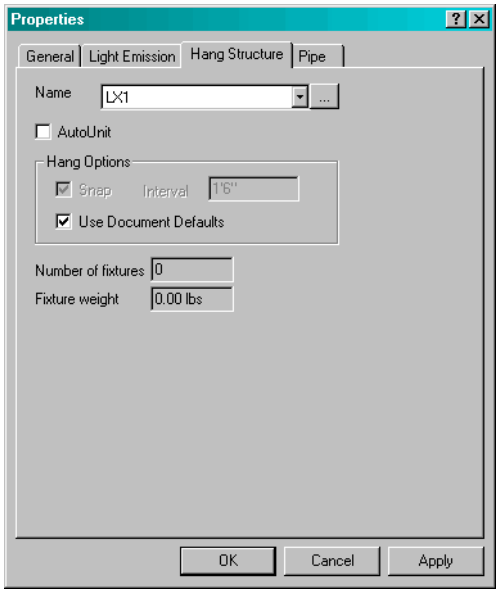
6 To reorder the position names in the list, select the name of the position you want to move. Click *Up* or *Down* to move the name in the list.

7 If you are in a plot view and want to determine what a particular position references, click *Select*. If the plot view is open (and the correct layers are marked editable), the hang structures that use that position name are selected. To select more than one position, press CTRL and then click on the position.

8 Click *OK* to close the Position Manager.

Hang structure properties

Options on the *Hang Structure* tab affect the properties of the selected hang structure(s).



a. *Name:* select a new name for the hang structure from the *Name* drop-down list. Click the ellipsis (...) button to open the Position Manager and create new names.

b. *AutoUnit:* select this check box to assign a number automatically to each fixture that you hang. Fixtures are automatically renumbered if you add a new fixture into the middle of a row of fixtures. This option is available for pipes only and does not apply to truss.

c. *Snap:* select this check box to hang fixtures on the pipe at a certain interval as specified in the *Interval* box.

d. *Interval:* enter the distance between fixtures in this box. This option is used for pipe snaps.

e. *Use Document Defaults*: select this check box to use the interval for pipe snap as defined on the *Object settings* tab in *Show Options*. For more information on this tab, refer to "Object Settings" on page 5.2-8.

f. *Number of fixtures:* number of fixtures on the hang structure.

g. *Fixture weight:* total weight of fixtures on the hang structure. The weight measurement accounts for fixture weight only and does not consider cable or other safety ratings.

Drawing pipes

Pipes are single member hang structures. Pipes can be set to draw from centre or from start.

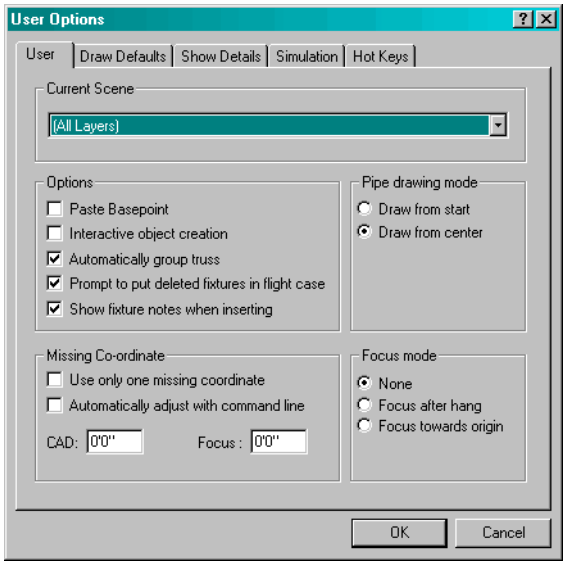
The first pipe drawing mode, Draw from Start, requires pipes to be drawn by specifying the start point and the end point (from one extremity of the pipe to the other).

The second pipe drawing mode, Draw from Center, requires that the pipe be drawn by specifying the centre point and one end point or extremety of the pipe. Although it is only necessary to enter all points when drawing in interactive mode, the pipe drawing mode will affect the insertion point of the pipe when drawing using the pipe dialog box. The drawing mode will also affect how the fixtures are hung when using pipe snap, how the fixture offset distance is measured, and how pipe tape prints are labelled.

To set the pipe drawing mode

1 From the **Options** menu, choose **User Options**.

2 Click the *User* tab.



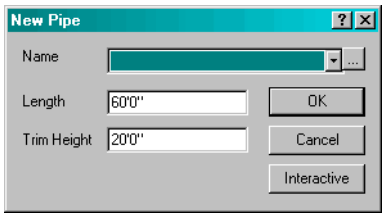
3 In the *Pipe Drawing Mode* section, select *Draw from Center* or *Draw from Start*.

Tips:

- You can change the default setting for a pipe in mid command by right-clicking and choosing the drawing mode for that pipe just before inserting it into the drawing.
- You can change the pipe mode for a specific pipe at any time in the pipe's properties. For more information, refer to "Pipe properties" on page 6.7-5.

To draw a pipe

1 From the **Draw** menu, choose **Pipe**.



2 In the *New Pipe* dialog box, find the name of the pipe. If you need to create the name of the pipe, click on the ellipsis button [...] to open the Position Manager. Click *New* and type in the name of the pipe.

3 In the *Length* box, enter the length of the pipe, if different from the default. The default value is 60'0".

4 In the *Trim Height* box, enter the desired trim height for the pipe. The default value is 20'0".

Note: Pipes ignore the missing coordinate setting when drawn using this method.

5 Click *OK*.

Tip:
You may also use the *Pipe* tool on the *Draw* toolbar.

To draw a pipe from center in interactive mode

1 From the **Draw** menu, choose **Pipe**.

2 Set the height of the pipe that you are about to draw by pressing the TAB key and entering a value for the missing coordinate if applicable.

3 Click to place the centre point of the pipe.

4 Drag it to the proper length and click to place the end point of the pipe. Notice the pipe extends in both directions - mirrored at center.

5 Right-click on the pipe you have just drawn and click **Properties**.

6 Click *Hang Structure* and scroll to find the name for this new pipe. If the name is not already in the list, use the *Position Manager* [...] to enter the new name.

7 Repeat to draw more pipes.

To draw a pipe from start in interactive mode

1 From the **Draw** menu, choose **Pipe**.

2 Set the height of the pipe that you are about to draw by pressing the TAB key and entering a value for the missing coordinate if applicable.

3 Click to place the starting point of the pipe.

4 Drag it to the proper length and click to place the end point of the pipe. Notice the pipe extends in one direction.

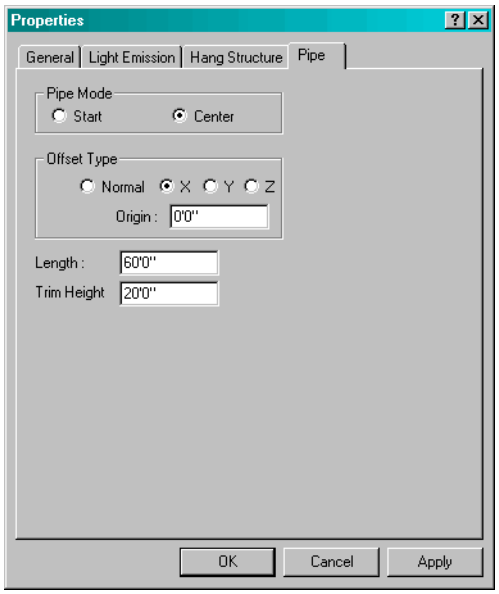
5 Right-click on the pipe you have just drawn and click **Properties**.

6 Click *Hang Structure* and scroll to find the name for this new pipe. If the name is not already in the list, use the Position Manager [...] to enter the new name.

7 Repeat to draw more pipes.

Pipe properties

Options on the *Pipe* tab affect the mode, offset, and dimensions of a pipe.



a. *Pipe mode*: select the pipe mode for the specific pipe. The available modes are:

Start. If enabled, pipes are drawn from one end to the other. When start mode is used, offset is measured from the beginning of the pipe in a positive direction.

Center. If enabled, pipes are drawn from the center out to both ends at the same time. When center mode is used, offset values are either positive or negative measured from the center of the pipe.

b. *Offset Type*: offset is a fixture property that measures where a fixture is hung on a pipe. The value will depend on the pipe mode selected.

Pipes can also base the Offset on a specified coordinate. In this case, a fixture's offset value is calculated as the distance on the specified axis from the specified coordinate. This is handy if you have many small pipes grouped together to form one hang position, for example the

Balcony Rail. If this is the case, choose all the pipes' properties to have their Offset based about X=0. Fixtures on the Balcony Rail will then report where they are relative to the theatre rather than the beginning of the pipe.

This offset methodology can be extended to box booms as well. For example, a boom in the first box (which is 10' above the deck) can be drawn as a vertical pipe and have its offset to be based about Z=10'. A fixture's offset will then tell you how high (from the bottom of the box) to hang it.

In WYSIWYG, set the offset mode that you want to use. The choices are:

Normal, which indicates that the offset is based on the pipe mode.

X, *Y*, and *Z*, which calculates the offset based on the specified coordinate.

If you have selected an X, Y, or Z offset type, you must specify a coordinate on the axis from where offset values will be calculated.

c. *Length*: enter new text to change the length of the selected pipe.

d. *Trim Height:* enter new text to change the trim height. If the pipe is vertical, this value will determine the lowest Z value of the pipe.

Hanging truss

Truss is a hang structure with multiple hanging members. Truss differs from pipes in that it is built from library objects instead of being drawn. WYSIWYG contains an extensive library of different truss types from many different manufacturers. The truss library also contains floor mounts, booms and ladders.

To hang truss

1 Open the Library Browser and click the *Truss* tool.

Tip: You can also click the *Truss* tool on the *Draw* toolbar to place truss.

2 Navigate through the library to find the type of truss you want to use.

3 Double-click on the object name.

Result: A piece of truss will attach to the cursor.

4 Click on the drawing to place the first piece of truss.

5 Continue placing pieces of truss as needed.

Note: To change the truss type, return to the library, double-click on the name of the new desired truss type and continue placing truss pieces as normal.You can also use Truss shortcuts.

6 Right-click and choose **Finish Placing Truss**.

7 Right-click on the truss you have just inserted and click **Properties**.

8 Click *Hang Structure* and scroll to find the name for this new truss. If the name is not already in the list, click the ellipsis button [...] to enter the new name.

To insert a shortcut for truss objects

Refer to "To create a new shortcut" on page 4.2-2.

Using Assembly snap with truss

Assembly snap will assemble truss pieces together as you insert them. Assembly snap ensures proper structural assembly. This is especially useful for long or complex structures made up of many individual pieces.

As with all snap functions (discussed in "Snaps" on page 6.2-6), Assembly snap can be selected in mid- command or as a running snap type.

To select Assembly snap as a running snap type

Enable Assembly snap before inserting truss to assemble and group all subsequently inserted pieces of truss.

To select Assembly snap in mid-command

Enable Assembly snap after inserting the first piece of truss to assemble selectively. Assembly snap will disengage after you place the next piece of truss, assembling only the first truss (or group of trusses) with the second piece.

To use Assembly snap

1 From the **Tools** menu, choose **Snap**, then choose **Assembly Snap**.

Tip: You can also click the *Assembly* snap tool on the *CAD Options* toolbar.

2 Insert truss as described above. To snap truss pieces together, hover the cursor over the end of the piece of truss you want to attach the next piece to and it will automatically snap into position.

6.8 Hanging and Focusing Fixtures

Fixtures are objects on the plot and appear in 3D in shaded views. When you insert a fixture into the plot, it must be hung from a hang structure. WYSIWYG has an extensive library of fixtures to choose from. Inserting fixtures into your drawing is as easy as pointing and clicking. You can insert fixtures directly from the Library Browser, or you can create shortcuts to your favorite fixtures.

Hanging fixtures

To insert fixtures

1. From the **Library** menu, choose **Browse Library**.
2. Click the *Fixture* tool at the bottom of the browser.
3. In the tree menu that is now visible, click on the "+" sign beside *Manufacturer* to select fixtures by manufacturer name, *Type* to select by fixture type, or *All* to see all the fixtures in alphabetical order.
4. Double-click on the fixture name you want to insert.

   Result: A fixture with default settings for this type will attach to the cursor.
5. To change the properties of the fixture before inserting, right-click on the fixture name and click **Property**.

   Result: A dialog box will open with a shaded view of the fixture on the left and an image of its symbol on the right. Use the *Photometrics* tab to change the lamp and lens settings for the fixture, if applicable. For more information on the Photometrics tab, refer to "Photometrics" on page 6.8-11. Click *Insert* and the dialog will close.
6. Move the mouse over a hang structure and click to place the fixture. Continue placing this type of fixture by clicking on the other hang structures as desired.
7. Finish placing this fixture type by right-clicking and choosing **Finish placing fixtures** from the menu that appears.
8. To place other fixture types, repeat the above steps for each type.

To replace fixtures

1. Select the fixture(s) you want to replace.
2. Right click and choose **Replace Fixture**.
3. Select the new type of fixture desired from the fixture selection dialog box.
4. Click *Insert*.

   Result: All the fixtures that were selected in step 1 will be replaced with the fixture type chosen in step 3. Some fixture attributes may be modified as a result of the replacement.

Custom fixtures

To create a custom fixture

1. Custom fixtures must be created on the WYSIWYG welcome screen with all files closed. If you are currently working on a document save and close it.
2. On the WYSIWYG welcome screen, from the **Options** menu, choose **Create New Conventional**.

   Result: The *Fixture Wizard* opens.
3. Select the type of fixture you want to create.
4. Enter a name for the new fixture.
5. Click Next.
6. Select the circuit type where REGDIM is a regular dimmer.
7. Click *Next*.
8. Add or modify the path.

   Note: The path is the location in the library where the fixture will be stored. A fixture assigned to the path "Custom/Fresnels" would be available in the custom section of the Fixture library in a sub-category named FRESNELS.
9. Click *Next*.
10. Select the lamp base and default lamp for this fixture.
11. Click *Next*.
12. Specify the lens system for the fixture.
13. Click *Next*.
14. Enter a value in degrees for the beam angle for this fixture.
15. Enter a value in degrees for the field angle for this fixture.
16. Click *Next*.
17. Select the colour frame slot size.
18. Click *Next*.
19. Select a 2D symbol for this fixture from the symbol library.
20. Click *Next*.
21. Select a 3D symbol for this fixture from the shape library.

    Note: A 10" parcan is shown beside the selected shape in the viewing pane to offer a relative size measurement.
22. Click *Next*.
23. Enter the manufacturer, weight, and any other information.
24. Click *Next*.
25. Enter your name
26. Click *Finish*.

Note: Not all fields have to be completed. Fill in as much information as you can. You cannot delete or edit custom fixtures once saved.

Pipe snap

Pipe snap will place fixtures on a pipe at a specific interval. Pipe snap does not work for truss or other hang structures. It allows you to specify a pre-determined distance between fixtures.

As with all snap functions (refer to "Snaps" on page 6.2-6 for more information about snap tools), Pipe snap can be selected in mid command or as a running snap type.

To select Pipe snap as a running snap type

Enable Pipe snap before inserting fixtures to hang fixtures equidistantly.

To select Pipe snap mid-command

Enable Pipe snap after inserting the first fixture to enforce the interval selectively. Pipe snap will disengage after you place the next fixture.

To use pipe snap

1 From the **Tools** menu, choose **Snap**, then choose **Pipe Snap**.

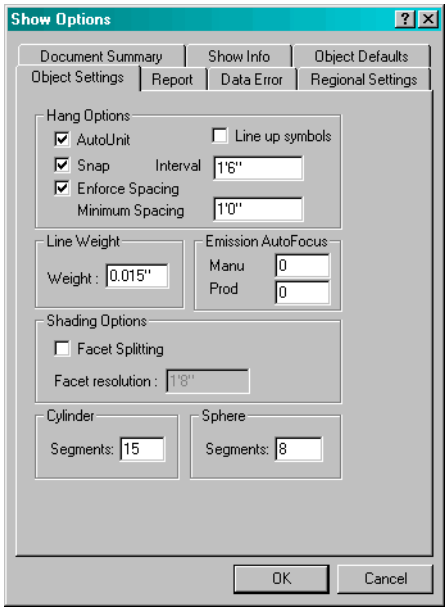
Tip: You can also click the *Pipe snap* tool on the *CAD Options* toolbar.

2 Insert fixtures as described above. With pipe snap active, the fixtures will automatically be hung at the pre-defined interval.

To set the snap interval

1 From the **Options** menu, choose **Show Options**.

2 Click the *Object Settings* tab.



3 Select the *Snap* check box if it is unchecked, and enter the new value in the *Interval* box.

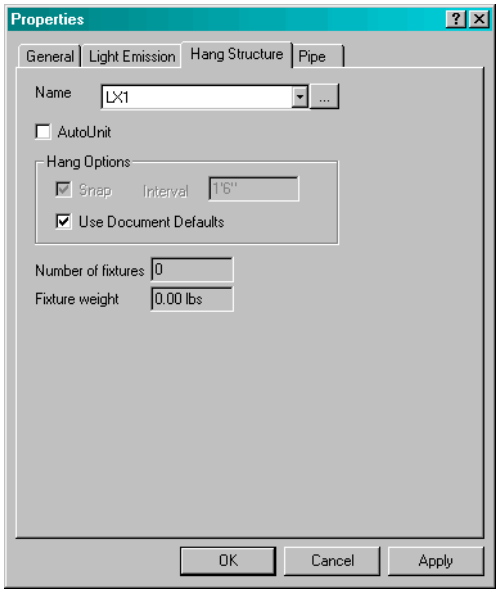
4 Click *OK*.

To set an interval for a specific pipe

1 Right-click the pipe.

2 Click **Properties**.

3 Click the *Hang Structure* tab.



4 Click to clear the *Use Document Defaults* check box.

5 Select the *Snap* check box.

6 Enter a new distance in the *Interval* box. This will be the measurement between fixture insertion points.

7 Click *OK*.

Focusing fixtures

You can focus the fixtures in your plot in three ways.

- You can click and drag the beam of a selected fixture.
- You can enter the pan, tilt, and spin values for the fixture in its *Properties* box.
- You can assign a focus position to the fixture.

To focus using click and drag

1 Select the fixture you want to focus.

Result: The beam will be shown.

2 Click on the beam and drag it to its new position. When you are done, release the mouse button.

Tips:

- You can also right-click the fixture and click **Focus** to drag the beam without having to hold the mouse button down continuously. Click to finish when the beam is focused where you want it.
- Use the missing focus coordinate to set a value on the inactive axis.

Drawing focus positions

Focus positions are points which fixtures can be focused at. Focus positions are objects in the drawing.

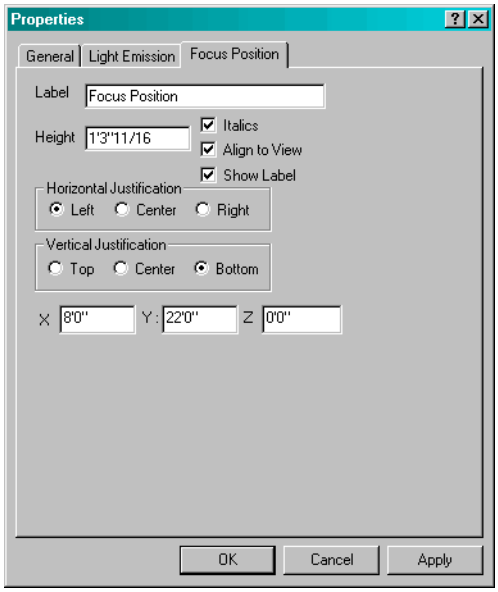
To insert a focus position

1. From the **Draw** menu, choose **Focus Position**.

   Tip: You can also click the *Focus Position* tool on the *Draw* toolbar to place focus positions.
2. Enter a label for the focus position.
3. Move the mouse so that the intersection of the crosshairs is placed where you want the focus position.
4. Click to insert the focus position.
5. Repeat these steps to insert all focus positions.

Focus position properties

Options on the *Focus Position* tab affect the alignment and style of the selected focus position.



The properties that can be modified include the justification, height, and presentation of a focus position. For a description of the justification and presentation-type properties, refer to “Text labels” on page 6.5-8.

Focus positions are unique in that you can adjust the focus position coordinates without needing to actually move the focus position on the drawing. The X,Y, and Z coordinate is the coordinate of the location in 3D space where the focus position is to lie.

Quick focus

Quick focus assigns a focus position or Pan and Tilt angle to the selected fixture.

To assign a focus position

1. From the **Tools** menu, choose **Quick Focus**.

   Tip: You can also click the *Quick Focus* tool on the *Tools* toolbar.
2. Click on the desired focus position. This will set that position as the destination for the next fixtures you select.

3 Click on each fixture you want focused to that position.

4 Right-click anywhere in the wireframe view and choose **Finish Quick Focus**.

To assign a pan and tilt angle

1 From the **Tools** menu, choose **Quick Tools**.

2 In the dialog box that opens, check *Focus*.

3 Enter a value in degrees for *Pan* and *Tilt*.

4 Click *OK*.

5 Click on each fixture you want oriented to the specified angle.

6 Right-click anywhere in the wireframe view and choose **Finish Quick Tools**.

To focus from the fixture's properties

1 Select the fixture(s) you wish to focus.

2 Right-click and choose **Properties**.

3 Click the *Fixture* tab.

4 Enter values in degrees for *Pan*, *Tilt*, and *Spin* or select a focus position from the pull-down list.

Note: For the definitions of these attributes, please refer to "Data fields" on page 7.2-1.

5 Click *OK*.

Fixture-specific focusing tools

All fixture-specific focusing tools such as shuttering, spotting/flooding, and lens changes are available through the selected fixture's properties on the *Shutter*, *Lens*, and *Fixture* pages respectively.

To make shutter cuts

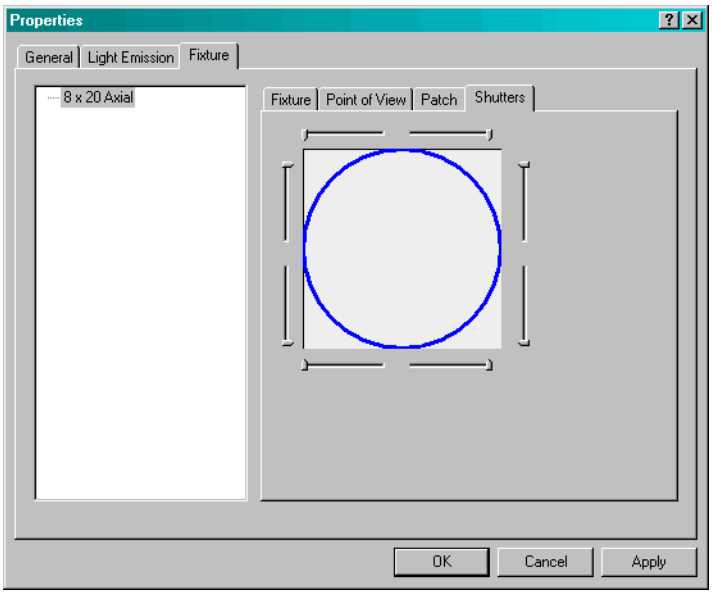
1 Select the fixture(s) for which you want to make shutter cuts.

2 Right-click and choose **Properties**.

3 Click the *Fixture* tab.

4 Click the *Shutters* tab.

Note: The POV image is not available when a fixture is zoomable or articulable.



5 Use the sliders along the top, bottom, and the sides to move the shutter blades.

Result: The cuts that you make are visible in the drawing.

6 Click *OK* when finished.

To adjust the leaves of barndoors

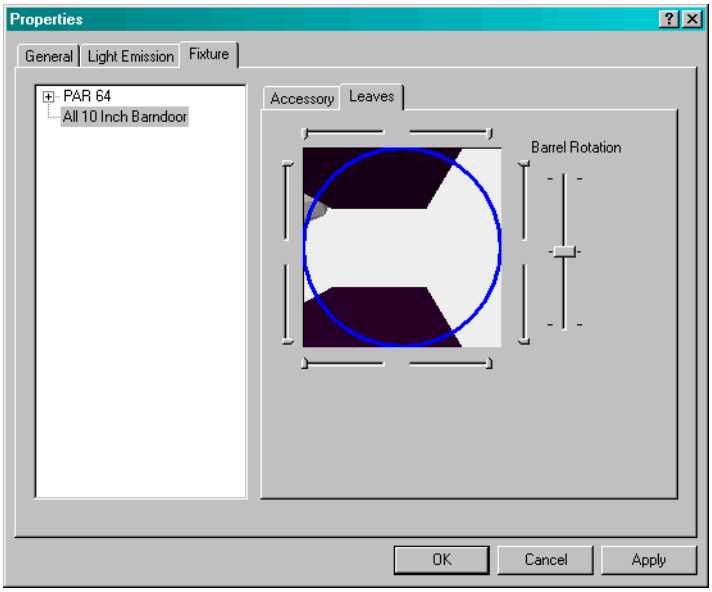
1 Select the fixture with barndoors for which you want to adjust the leaves.

2 Right-click and choose **Properties**.

3 Click the *Fixture* tab.

4 Highlight the barndoor from the list on the left side.

5 Click the *Leaves* tab.



6 Use the sliders along the top, bottom, and the sides to move the barndoor leaves. You will notice that the movement of the leaves is restricted to left, right, up, and down.

Result: The cuts are visible in the drawing.

7 Click *OK* when finished.

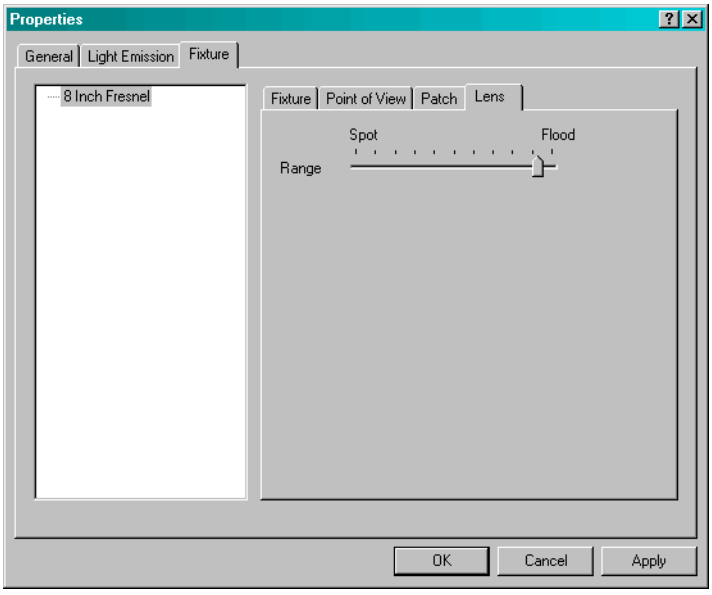
To adjust flood or spot

1 Select the fixture(s) for which you want to adjust the flood or spot.

2 Right-click and choose **Properties**.

3 Click the *Fixture* tab.

4 Click the *Lens* tab.



5 Use the slider to adjust the spot and flood.

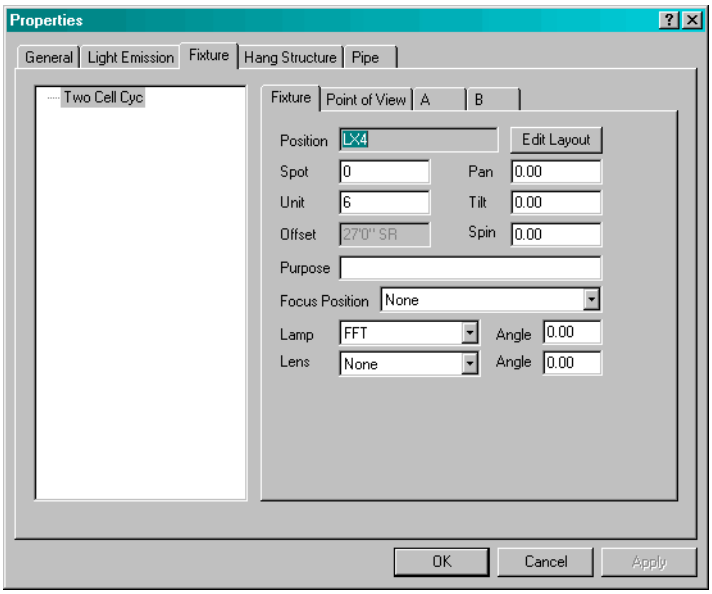
6 Click *OK* when finished.

To focus a Par

1 Select the PAR fixture that you want to focus.

2 Right-click and choose **Properties**.

3 Click the *Fixture* tab.



4 From the *Lamp* drop-down list, select the lamp that you want to focus.

5 In the *Angle* box for the *Lamp*, enter the rotation angle of the lamp. The angle determines the degree of rotation of the lamp within the housing.

6 Click *OK* when finished.

Fixture properties

All fixtures in WYSIWYG have properties. Properties are the characteristics or attributes that shape and define a fixture.

Fixture properties can be divided into two categories:

- the properties that are common to all fixtures in WYSIWYG
- the properties that you define by assigning attributes to a fixture.

Fixtures also have many non-visible attributes, some of which can be edited. These include Alias (found as Type in the Reports), Cost, Weight and AutoFocus codes.

General fixture properties

In WYSIWYG there are four tabs that appear in the properties dialog box for every fixture. These tabs are the Appearance, Version, Data, and Photometric tabs.

All four tabs are explained here.

To view fixture properties that are common to all fixtures

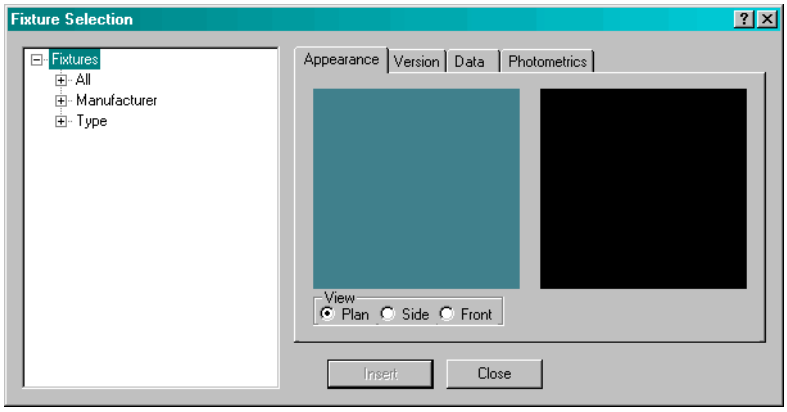
1 From the **Library** menu, choose **Browse Library**.

2 Click the *Fixture* tool at the bottom of the browser.

3 Right-click on the fixture name and click **Property**.

Tip: You can also click the *Property* tool at the top of the Library Browser.

Result: The *Fixture Selection* dialog box is displayed.



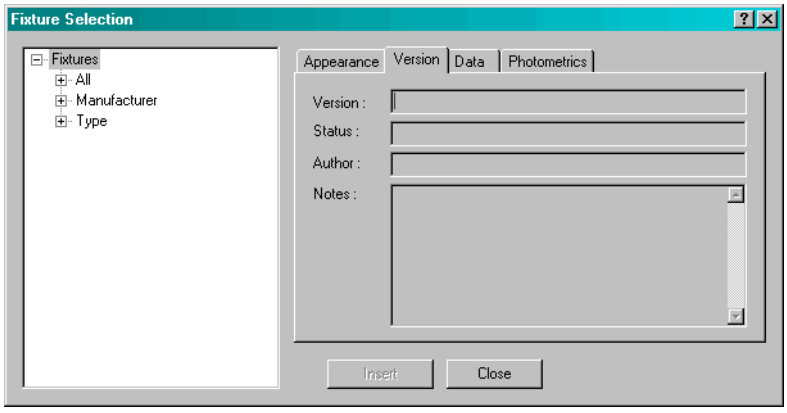
4 Select a fixture from the tree menu on the left side.

Result: A shaded view and an image of the fixture's symbol are displayed in the two boxes on the right side.

5 Click *Plan*, *Side*, or *Front* to view the fixture in the different views. The Wireframe view changes depending on the view type that you select.

1 Version

Options on the *Version* tab are read only and are set when the fixture is added to the library.



a. *Version*: release number of the fixture.

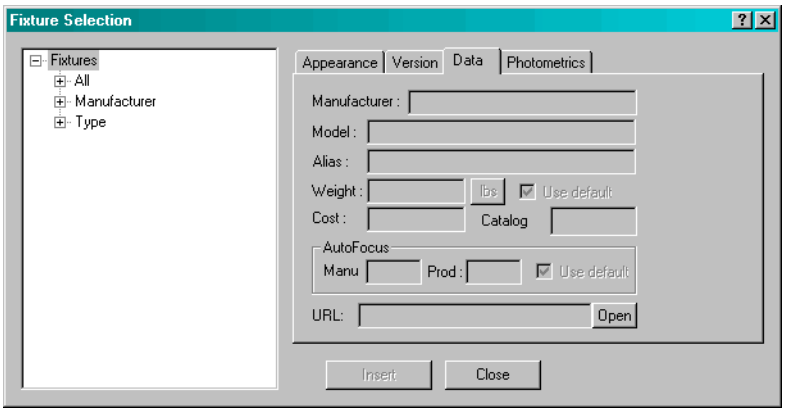
b. *Status*: status of the fixture. For example, *Beta* indicates the fixture is completed but not verified. *Release* indicates the fixture has been tested and is accurate. A status of *Preliminary* indicates that the fixture has minimal functionality.

c. *Author*: the individual or company who constructed the fixture.

d. *Notes*: any notes that pertain to the fixture. The notes usually pertain to the simulation capabilities in WYSIWYG.

2 Data

Options on the *Data* tab are read only and are set when the fixture is added to the library. This does not, however, include the *Alias*, *Weight*, *Cost*, *Manu*, and *Prod* fields.



a. *Manufacturer*: name of the manufacturer of the fixture.

b. *Model*: model number of the fixture.

c. *Alias*: alternate name for the fixture.

d. *Weight*: weight of the fixture.

e. *Use default*: select this check box to indicate that the default weight is to be used.

f. *Cost:* cost of renting or purchasing the fixture. The default value is 0.00.

g. *Catalog*: code that is used by the manufacturer to identify the fixture.

h. *Manu*: manufacturer's code for those fixtures that are Autofocus-compatible. Conventional fixtures do not have a manufacturer's code. Default: 127

i. *Prod*: product code for those fixtures that are Autofocus-compatible. Conventional fixtures do not have a product code. The default value is 127.

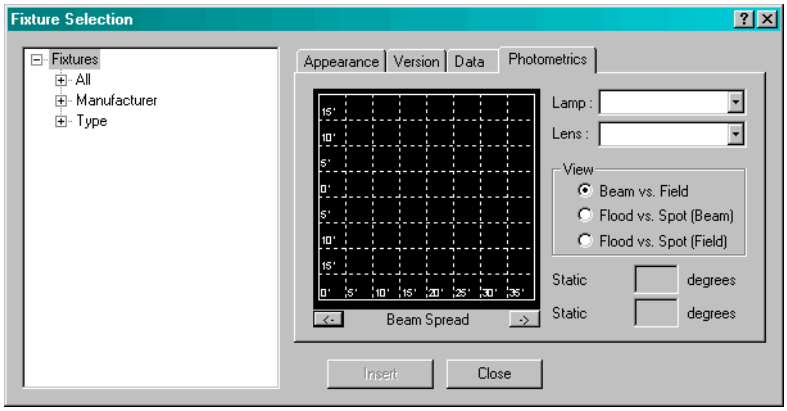
j. *Use default*: select this check box to indicate that the default settings for Prod and Manu (127, 127) are to be used.

k. *URL*: Internet address of the manufacturer.

l. *Open*: opens the Web page of the manufacturer.

3 Photometrics

Options on the *Photometrics* tab affect the lamp and lens settings for the fixture.



a. *Lamp*: select the lamp to be used in the fixture.

b. *Lens*: select the lens setting for the fixture from the *Lens* drop-down list. Changing the lens alters the photometric properties of the fixture.

c. *View*: select the graphical representation that you want to display in the box to the left. The choices are as follows:

Beam vs. Field. Displays a graphical representation of the beam and field angles of the fixture.

Flood vs. Spot (Beam). Displays a graphical representation of the ratio between the flood and spot for the fixture's beam angle.

Flood vs. Spot (Field). Displays a graphical representation of the ratio between the flood and spot for the fixture's field angle.

d. *Field angle*: measurement in degrees from the centre of the beam to where the light level drops off to 10% of the peak.

e. *Beam angle*: measurement in degrees from the centre of the beam to where the light level drops off to 50% of the peak.

Fixture-specific properties

Customize the properties of a fixture by assigning specific attributes such as point of view, patch information, and lens type.

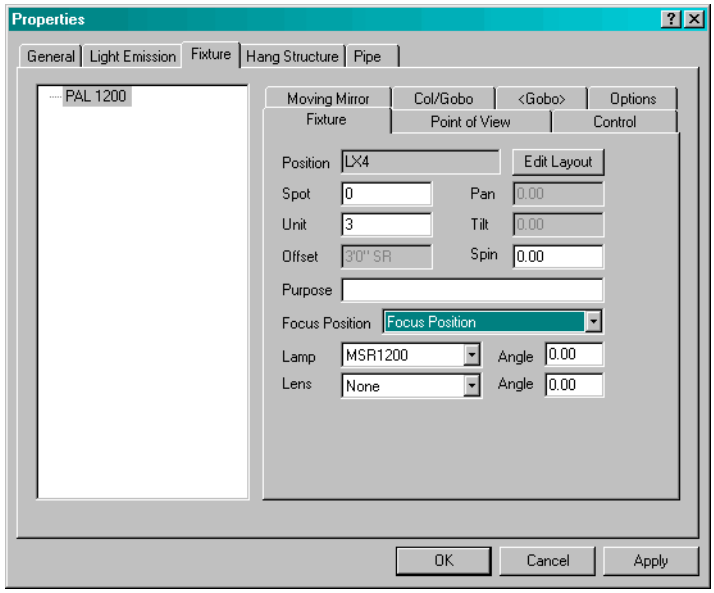
To edit/ view fixture attributes

1 Select the fixture for which you want to view or edit.

2 Right click and select **Properties**.

Tip: At any time, to access an object's properties, you can click on the *Properties* tool on the *Edit* toolbar. P

Result: The *Properties* dialog box is displayed.



1 Fixture

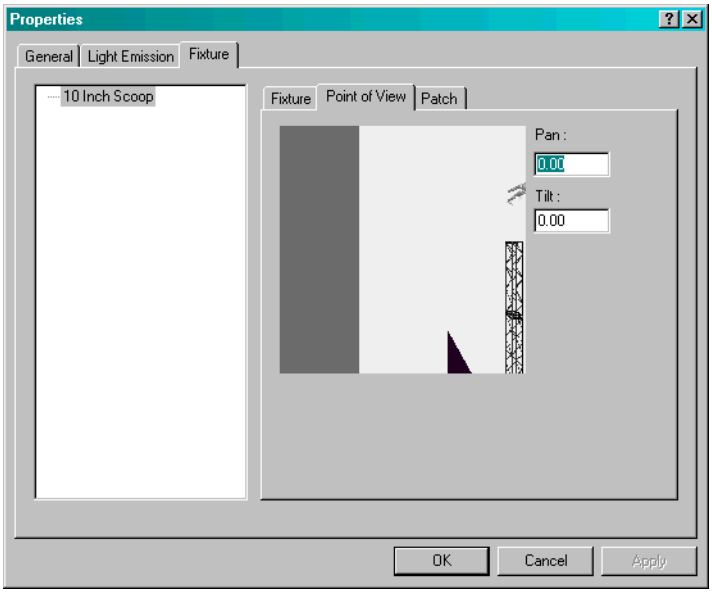
Options on the *Fixture* tab affect the fixture properties and their accessories. On the left side of the dialog box is a list of the selected fixtures and their accessories. Click on a list item to obtain further details about the item. Refer to "Data fields" on page 7.2-1 for a description of the fields.

a. *Edit Layout:* click to edit the text label layout for the selected fixture. For more information on layout refer to "Information layout" on page 6.8-17.

2 Point of view

Options on the *Point of View* sub-tab affect the orientation of the fixture. Use the arrow keys to pan and tilt the light.

The POV sub-tab is not available if a fixture is articulable (that is, if any accessories in the fixture tree have a moving head or moving mirror modifier).

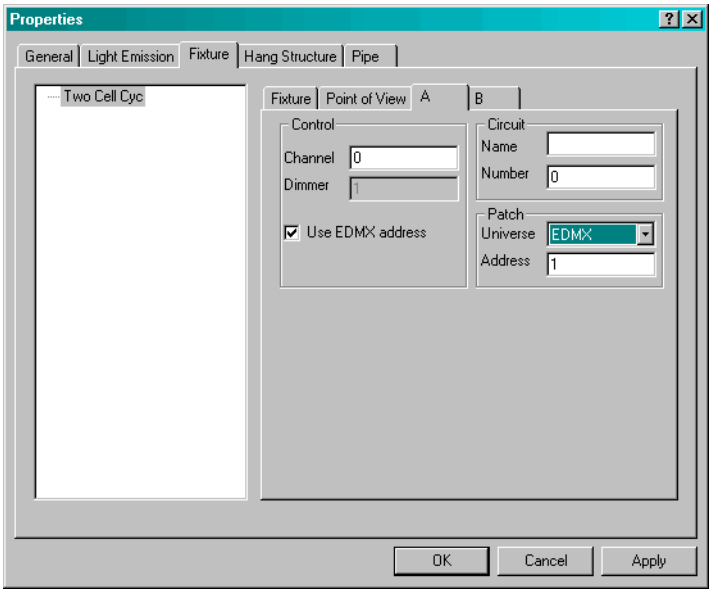


a. *Pan*: enter a value for the pan and view the result in the window to the left.

b. *Tilt*: enter a tilt value and view the result in the window to the left.

3 Patch

Options on the *Patch* sub-tab affect the circuit, channels, and patch universe for the selected fixture.

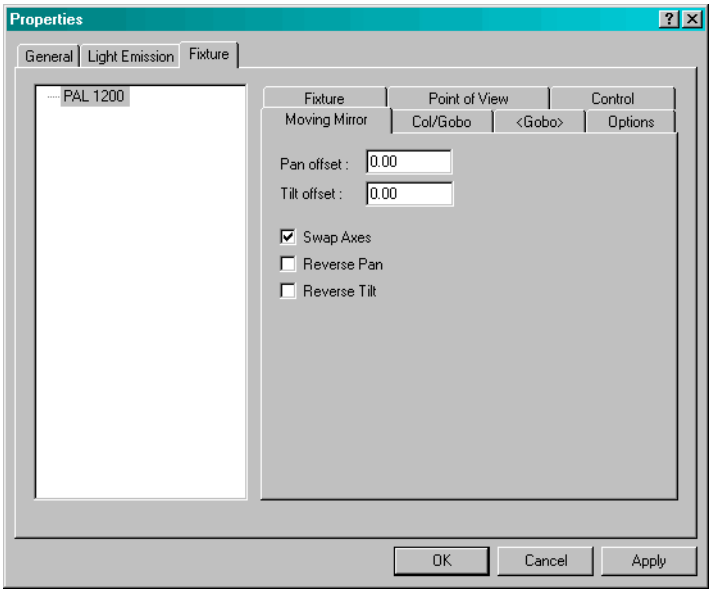


a. *Control, Circuit, and Patch*: refer to "Data fields" on page 7.2-1 for a description of the fields.

b. *Use EDMX address:* select this check box to use the EDMX address for the patch universe. This check box is enabled if EDMX is selected from the *Universe* drop-down list. EDMX is an Ethernet protocol developed by ETC.

4 Moving mirror/moving head

Options on the *Moving Mirror* and *Moving Head* sub-tabs affect the pan and tilt values.



a. *Pan offset*: sets the manually adjustable pan angle.

b. *Tilt offset*: sets the manually adjustable tilt angle.

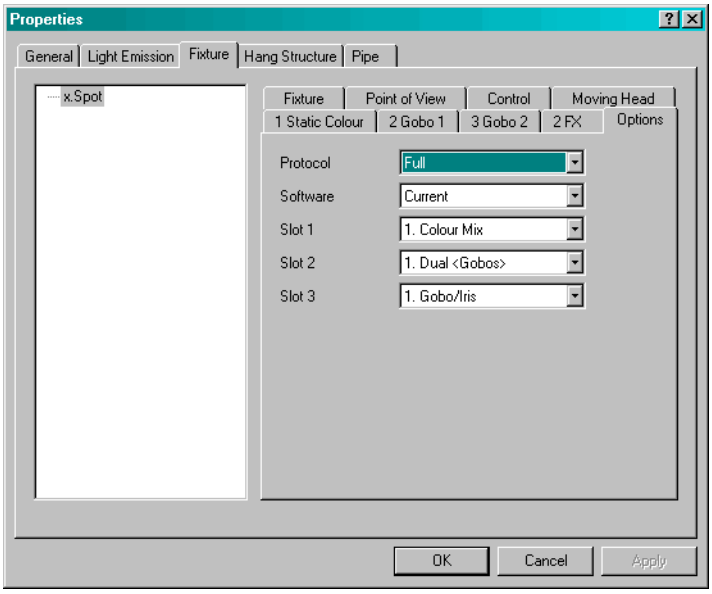
c. *Swap Axes:* select this check box only if the real fixtures have an option to swap the control channels for the tilt and pan and that option is set to true.

d. *Reverse Pan:* select this check box only if the real fixtures have an option to reverse the pan parameter and that option is set to true.

e. *Reverse Tilt:* select this check box only if the real fixtures have an option to reverse the tilt parameter and that option is set to true.

5 Options

Options on the *Options* tab affect the mode selection for the fixture(s).



a. *Mode*: sets the mode for the selected fixture.

b. *Software:* software version of the software that is loaded into the selected fixture.

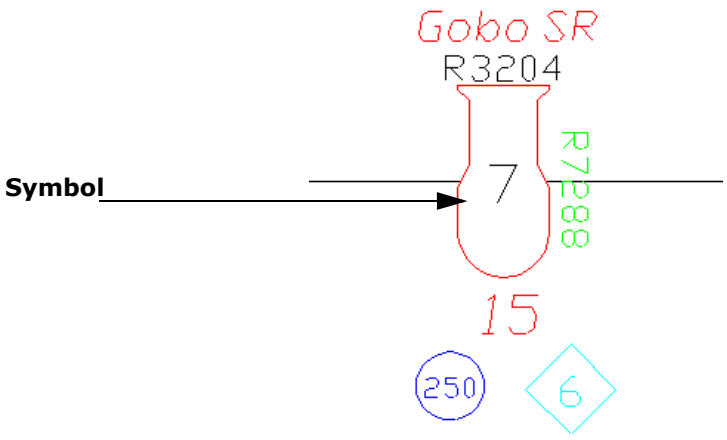
c. *Slots*: mechanical components that are present in the selected fixture.

d. *Control:* simulates the lamp control channel on the selected fixture. Since you can physically turn a fixture on or off, WYSIWYG enables this action in the software.

Fixture notation

Fixture notation affects how fixtures and their properties are displayed on your drawing and consequently on your plot in Presentation mode. As illustrated below, two components comprise fixture notation:

- symbol
- information layout.



Symbols

A symbol is a graphic representation of a fixture in a Wireframe view. In all wireframe views except isometric, symbols may be used to represent fixtures instead of three dimensional models. Fixtures are associated to specific symbols by default, but you can change the symbol to whatever you like.

To turn fixture symbols on or off

1 From the **Options** menu, choose **User Options**.

Result: The *User Options* dialog box is displayed.

2 Select the *Show Details* tab.

3 Select the *Draw Symbols* check box.

4 Click *OK*.

To change the symbol for a fixture type

The following procedure will change the symbol for all instances of the selected fixture type in the document. Similar procedures will change the symbol at an application level or selected-fixture-only level.

Note: For fixtures with interchangeable lenses (or lamps in the case of PARs), you must change the symbol for the lamp or lens instead of the fixture.

1 From the **Library** menu, choose **Browse Library**.

Note: To change the symbol at an application level, you must browse the library from the WYSIWYG welcome window.

To change the symbol for the selected fixture only: select the fixture, open its properties and skip to step 5.

2 Select the Fixture library.

3 Navigate through the library and select the fixture type for which you wish to change the symbol.

4 Click the *Property* tool.

Result: The properties dialog for that fixture type will open.

5 Click *Change Symbol*.

Result: The symbol selection window is displayed.

6 Select the new desired symbol.

7 Click *Select*.

8 Close the fixture property box.

Information layout

The following attributes are available to be placed around fixture symbols in WYSIWYG: focus, lamp, lens, notes, position, purpose, spot, unit, channel, circuit, colour, gobo, patch, and dimmer.

The position, visibility, colour and justification of these attributes can be set by editing the Label Layout.

Just like symbols, label layouts can be edited at an application level, document level, or selected-fixture-only level.

To change the Label Layout

The following procedure will change the label layout for all instances of the selected fixture type in the document. Similar procedures will change the symbol at an application level or selected-fixture-only level.

1 From the **Library** menu, choose **Browse Library**.

Note: To change the label layout at an application level, you must browse the library from the WYSIWYG Welcome window.

Note: To change the label layout for the selected fixture only, select the fixture, open its properties and skip to step 5. If more than one fixture type is selected the edit layout option will not be available.

2 Select the Fixture library.

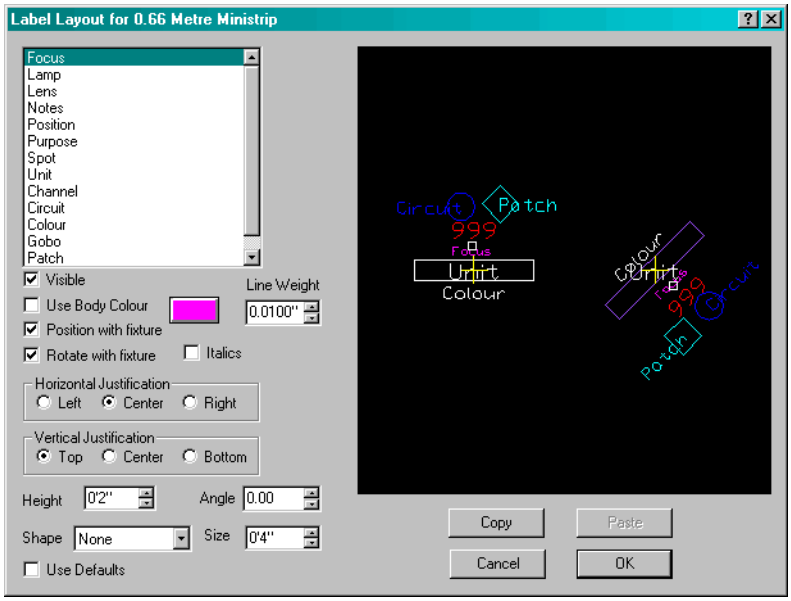
3 Navigate through the library and select the fixture type for which you wish to change the label layout.

4 Click the *Property* tool.

Result: The properties dialog box for that fixture type is displayed.

5 Click *Edit Layout*.

Result: The *Label Layout* window is displayed.



6 Select the attribute you wish to edit from the list.

Note: To make the controls active you may have to click to clear the *Use Defaults* check box.

7 Modify the individual properties.

Note: When you make a change to an attribute, the results are previewed in the window on the right in the Edit Layout Dialog.

a. Select the *Visible* check box if you want the attribute to be displayed around the fixture symbol in your plot.

b. Select the *Use Body Colour* check box if you want the attribute label to use the same font colour as the fixture symbol. Leave the box unchecked and choose a colour to set a specific colour for that attribute.

c. Adjust the line weight to increase or decrease the density of the label font.

d. Select the *Italics* check box to italicize the label font. Note there are no options to change the font.

e. Select the *Position with Fixture* check box if you want the attribute's label to maintain its position in relation to the fixture symbol regardless of where the fixture is focused.

f. Select the *Rotate with Fixture* check box if you want the attribute's label to maintain it's alignment in relation to the fixture symbol regardless of where the fixture is focused. Note this option is only available if *Position with Fixture* is selected.

g. Set the horizontal and vertical justification for the text. This is in relationship to the insertion point or shape.

h. Increase or decrease the height to modify the font size.

i. Set the angle to specify the label alignment in relationship to the fixture symbol.

j. Select a shape to be associated with this attribute if desired. You can choose from: none, circle, square, octagon, or diamond.

k. Increase or decrease the size of the shape as desired.

8 Set the position of the label by clicking and dragging on the label in the preview window.

Note: The preview window shows the fixture symbol and layout in two orientations for your reference. This is to give you a visual for the fixture in its unfocused and focused positions.

9 Repeat the procedure for all attributes.

Tip:
The copy and paste buttons allow you to apply the same label layout to other fixture types.

Inserting symbols

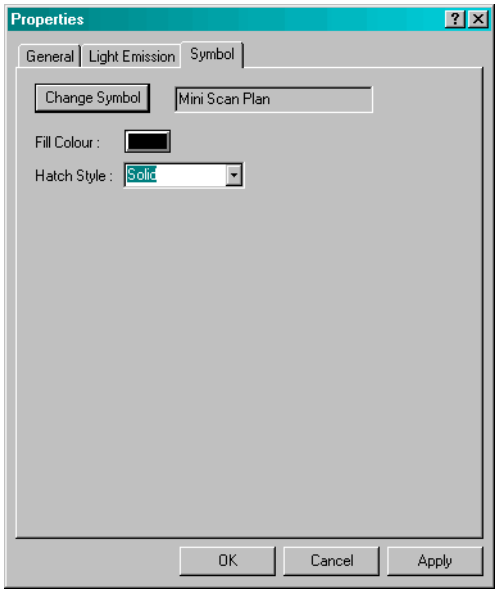
You can also insert symbols that are not associated to fixtures. This allows you to represent the fixtures when you draw a key or want to show the position of a boom in a plan view. Symbols are not fixtures and therefore will not be counted in reports. Standard symbols, such as the United States Institute for Theatre Technology (USITT) symbols are available from the library for your use.

To insert symbols

Please refer to "The Library Browser" on page 6.3-1. Inserting objects from the library is discussed in "To insert objects from the library" on page 6.3-2.

Symbol properties

Options on the *Symbol* tab affect the fill colour and hatch style of a symbol.



a. *Change Symbol*: click to select a different symbol.

b. *Hatch Style*: the style to use for the hatch. If the Hatch style is *None* the symbol is transparent. If it is *Background* , the symbol will fill in with the colour of the background and will not be transparent.

c. *Fill Colour*: the colour with which you want the symbol to be filled. If the hatch style is *None* or *Background*, the fill colour is not enabled.

6.9 Colour, Gobos, and Accessories

You can add accessories to fixtures such as barndoors, tophats, colour scrollers, dousers, etc. Accessories have their own section in the Library Browser and can be inserted just like other objects. Accessories must be inserted onto existing fixtures and can only be inserted onto compatible fixtures.

To insert an accessory

1. Open the Library Browser.
2. Click on the Accessory section tool.
3. Find the accessory you want to insert.
4. Double-click the accessory name.
5. Click on the fixture(s) to insert the accessory.

Tips:

- To place an accessory on a selected fixture, right-click on the fixture and click **Add Accessory**. Use the browser to find the accessory you want to insert. Follow this procedure in Data mode and only those accessories that can be applied to the selected fixture will be available.
- To place an accessory on a group of selected fixtures, use the Accessory library. With the fixtures selected, find the accessory in the Library Browser. Right-click on the accessory name and click **Insert**.

To create a new shortcut to an accessory

Refer to "To create a new shortcut" on page 4.2-2.

Placing colour and gobos

You can drop colour and gobos onto fixtures by using the Library Browser. Colours and gobos from many manufacturers are included in the library, and you can preview the colour or gobo by double-clicking on the name.

To insert colours and gobos using the Library Browser

1. Open the Library Browser.
2. Click the Colour or Gobo section tool.
3. Find the colour or gobo you want to insert.
4. Right-click the colour/gobo name and click **Insert**.

   Result: The cursor will show a "Q".
5. Click on the fixture(s) you want to add the colour or gobo to.
6. Right-click and click **Finish Quick Tools** to finish placing the colour or gobo.

To create a new shortcut to a colour or gobo

1. Click the *Col/Gobo* shortcut bar and right-click in the open space on the bar.
2. Choose **New Shortcut**.

   Result: The *Colour Select or Gobo Select* dialog box is displayed.
3. Select a colour or gobo from the tree menu on the left side.

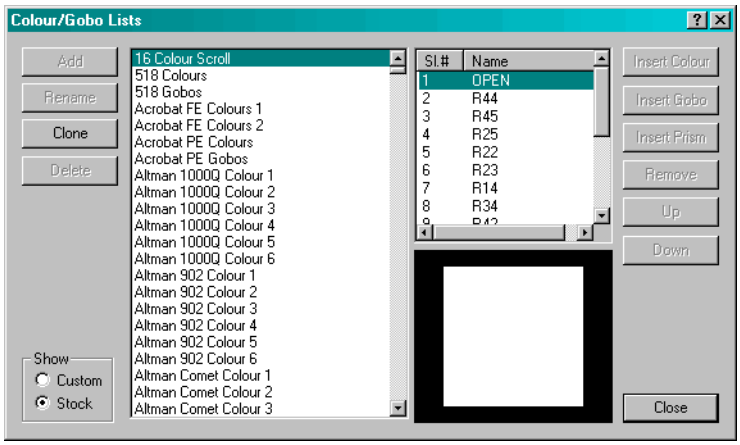
   Result: The colour or gobo is displayed on the right side.
4. Click *Select*.

Colour/gobo lists

Colour and Gobo lists are required for automated lighting fixtures and scrollers. By default, when inserted, automated fixtures and scrollers are set to use the manufacturer default colour or gobo list. WYSIWYG contains a library of "stock" colour and gobo lists provided by the fixture manufacturers. If you are using a custom set of colours or gobos, or want to generate a scroll for a colour scroller, you will need to create a custom colour or gobo list.

To view an existing library list

1. From the **Library** menu, choose **Edit Colour/Gobo Lists**.



2. In the *Show* section, click *Stock*.
3. Scroll to find the list you want to view.
4. Click on the list name. A list of slots and the colour or gobo for each slot is displayed in the slot window. If *OPEN* appears in the Name column for a slot, the colour, gobo, or prism has not been assigned or is intentionally left *OPEN*.
5. Click on a slot to see the preview of that colour or gobo in the box below the slot window.
6. Click *Close* when you are done.

To create a custom colour/ gobo list

1. From the **Library** menu, choose **Edit Colour/Gobo Lists**.
2. In the *Show* section, click *Custom*.

3 Click *Add*.

4 Enter the name of the new list.

5 Enter the number of slots on the wheel or scroll.

6 Click *OK*.

7 Click on the slot you wish to edit.

8 Click *Insert Colour*, or *Insert Gobo*, or *Insert Prism*. Make sure if you are inserting different types of media into the same list that this is actually possible for that fixture or accessory.

9 Click on the item you want to insert and click *Select*.

10 Repeat for the remaining slots.

11 To reorder the items in the slots, click on the slot and click *Up* or *Down* to move that item to a new position.

12 To erase a colour or gobo selection from a slot, click *Remove*. The slot returns to the default selection of *OPEN*.

13 Click *Close* when you are done creating lists.

Tip: It is recommended to clone the stock wheel or list associated with the fixture(s) and edit the cloned list. The cloned list inherits the properties of the original list.

To clone a stock colour/ gobo list

1 From the **Library** menu, choose **Edit Colour/Gobo Lists**.

2 In the *Show* section, click *Stock*.

3 Scroll to find the list you want to clone.

4 Click *Clone*.

5 Enter a name for the cloned list and click *OK*.

Result: The cloned list appears in the custom lists.

To rename a colour/ gobo list

1 From the **Library** menu, choose **Edit Colour/Gobo Lists**.

2 In the *Show* section, click *Custom*.

3 Select the colour/gobo list that you want to rename.

4 Click *Rename*.

5 Enter the new name for the list and click OK.

Result: The list is renamed.

To delete a colour/ gobo list

1 From the **Library** menu, choose **Edit Colour/Gobo Lists**.

2 In the *Show* section, click *Custom*.

3 Select the colour/gobo list that you want to remove.

4 Click *Delete*.

Result: The list is removed from the colour/gobo list.

To insert custom scrolls into scrollers

1. Select the fixture(s) with scroller(s) for which you want to assign a custom scroll.
2. From the **Edit** menu, choose **Object Properties**.
3. Select the *Fixture* tab.
4. In the fixture list, highlight the scroller.
5. Select the *Scroll* tab.
6. Select the desired list from the pull-down list.

   Note: Only custom list which have the appropriate number of slots and media type will be available.
7. Click *OK*.

To insert custom wheels into automated fixtures

1. Select the fixture(s) for which you want to assign a custom wheel.
2. From the **Edit** menu, choose **Object Properties**.
3. Select the *Fixture* tab.
4. Select the appropriate tab (ex. Colour 1, Gobo, <Gobo>).
5. Select the desired list from the pull down menu.

   Note: Only custom list which have the appropriate number of slots and media type will be available.
6. Click *OK*.

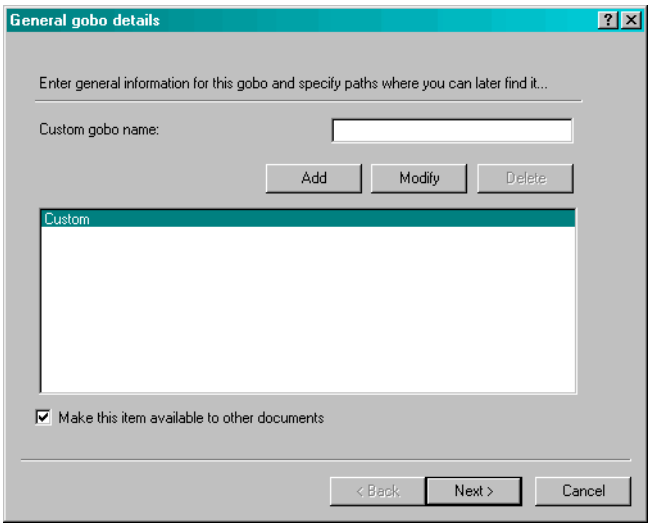
Custom gobos

You can generate custom gobos if you have the appropriate artwork in either bitmap or jpeg format. The Gobo Wizard walks you through the steps of making a custom gobo. Custom gobos are stored in the gobo library, under the Custom category once they are completed.

Custom gobos assigned to fixtures are only visible in shaded views and renderings. And then only the footprint is visible. Custom gobos do not break up the cone of light. In wireframe views, a substitute gobo is used in lieu of the bitmap or jpeg image. Custom gobos require an association with an existing WYSIWYG gobo in order to be visible in non-Open GL views.

To create a custom gobo

1 From the **Library** menu, choose **Gobo Wizard**.



2 Enter the custom gobo name.

3 Add or modify the path.

Note: The path is the location in the library where the gobo will be stored. A gobo assigned to the path "Custom/Corporate_Logos" would be available in the custom section of the gobo library in a sub-category named Corporate Logos.

4 To make this gobo available to other documents, check the appropriate box.

5 Click *Next*.

6 Type the path to the custom artwork file or use the browser [...] to open the image. The image file must be in either bitmap or jpeg format.

Result: A preview of the image will appear in the pane on the right and the image's attributes will be recorded in the box on the left.

7 Click *Next*.

Result: A short description of the next step appears. You will need to choose an existing library gobo to be displayed when the custom artwork cannot.

8 Click *Next*.

9 Navigate the library to find a suitable association gobo.

Note: Generic association gobos are stored in the Proxy category.

10 Select the desired gobo.

11 Click *Finish*.

Result: The custom gobo is placed in the gobo library as per the path specified in step 3.

Note: If your image file is not a perfect square (where the horizontal dimension is equal to the vertical dimension), your gobo projection will appear distorted. WYSIWYG is unable to maintain the aspect ratio. To avoid this problem, place your image within a black square and use the resulting image to generate your custom gobo.

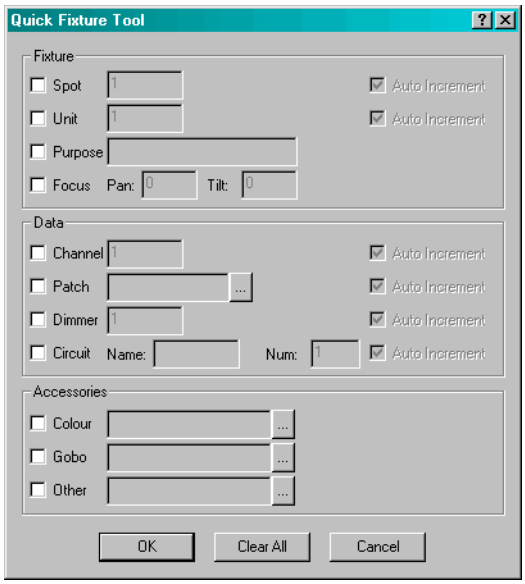
Quick tools

Quick tools allow you to quickly assign data, accessories, and other attributes to fixtures without having to switch to the Data mode or open the Properties box for each individual fixture. Quick tools will enter repetitive information, as well as incremental information for each fixture as you click on it.

To use Quick Tools

1 From the **Tools** menu, choose **Quick Tools**.

Tip: You can also click the *Quick Tools* tool on the *Tools* toolbar.



2 To activate a tool, select the check box to the left of the tool. You can activate as many as you want at a time. See below for explanations of the different tools.

3 Enter the initial value for the tool in the box to the right of the tool name. If you want the value to auto increment, as for channels or spot numbers, make sure you select the *Auto-increment* check box. WYSIWYG will assign the next sequential number based on the properties and requirements of the previous fixture.

4 When the dialog box is setup the way you want it, click *OK*.

5 The cursor will show a "Q", indicating that Quick Tools have been activated. Click on each fixture in the order you want any auto-incrementing values to be assigned.

Note: The attributes to be assigned to the next fixture are logged in the bottom left hand corner of the status bar.

6 When complete, right-click and choose **Finish Quick Tools**.

Note: To change the values at any time, right click and choose **Change Options**.

Tip: Use the *Clear All* button to reset all the values and deselect all quick tools.

Quick Tools

- Quick Spot enters a spot number in the *Spot* field.
- Quick Unit enters a unit number in the *Unit* field.
- Quick Purpose enters a fixture purpose in the *Purpose* field.
- Quick Focus enters a focus position or pan and tilt angle for the selected fixture.
- Quick Channel enters a channel number in the *Channel* field.
- Quick Patch enters a patch universe assignment in the *Patch* field.
- Quick Dimmer enters a dimmer number in the *Dimmer* field.
- Quick Circuit enters a circuit name and number in the Circuit *Name* and *Num* boxes.
- Quick Colour enters a gel colour number for the selected fixtures.
- Quick Gobo enters a gobo number for the selected fixtures.
- Other enters an accessory type for the selected fixtures.

Notes:

1. When entering values, type the exact value or use the [...] browse button to open a selection dialog.
2. The entry in the patch field must fulfill the requirements of patch notation which is *universe.#.* For more information regarding patch notation refer to "Reading the patch" on page 7.3-2.
3. For more information about the data fields, refer to "Data fields" on page 7.2-1.

6.10 The Flight Case

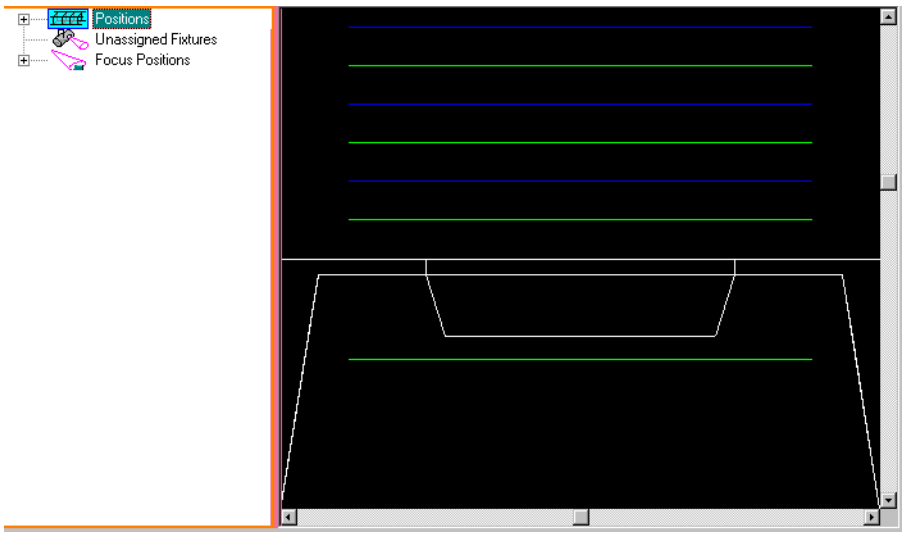
The Flight Case is a special view used to hold unhung fixtures, hang structures, and focus positions created or inserted in Data mode but not yet placed on the drawing. This is an ideal way to work from a set inventory as you can add all of your fixtures in Data mode and then click and drag the fixtures out of the Flight Case until you have exhausted your stock.

To open the Flight Case

1 From the **Tools** menu, choose **Flight Case**.

Note: The Flight Case may only be opened in a wireframe view.

Tip: The Flight Case layout is preconfigured with the flight case and a wireframe view.



How objects get into the Flight Case

As you enter fixtures in the Data mode spreadsheets, they appear in the Flight Case under the *Unassigned Fixtures* branch. When you enter position names into the Position Manager, the individual positions appear under the *Position* branch. As you assign fixtures to those positions, they will move from the *Unassigned* branch to under the proper position. All fixtures that appear in the Flight Case are unhung fixtures which means they are not in your drawing (regardless of whether they have been assigned a position or not).

If you create focus positions in the Data mode by typing labels in the focus field of the spreadsheet, those focus positions will appear under the *Focus Positions* branch of the Flight Case. Unhung fixtures assigned to those focus positions will appear under their respective Flight Case headings.

To learn more about inserting objects in Data mode, please refer to Chapter 7.0, “Data Mode.”

Entering and modifying objects in the Flight Case

You can insert fixtures and focus positions directly into the Flight Case using the Library, *Draw* menu or appropriate tools on the *Draw* toolbar.

You can change a fixture's assigned position by selecting the fixture in the Flight Case and dragging it to another position within the Flight Case.

You can change a fixture's assigned focus position by selecting the fixture and dragging it between Focus Positions within the Flight Case.

Transferring objects from Flight Case to the drawing

The items that appear in the Flight Case can be transferred to your drawing by clicking and dragging.

To hang fixtures from the Flight Case

1. Open the Flight Case.
2. Click and hold on the desired fixture.
3. Drag the fixture into the wireframe view and place it on a hang structure.

   Note: If a fixture appears in the flight case under a position branch, that fixture must be hung on that position. Fixtures with no position or unassigned fixtures may be hung on any hang structure.
4. Release the mouse button.
5. Repeat for the remaining fixtures.

   Tip: You can hang all the fixtures on one position by dragging the entire position out of the Flight Case.

   Selecting a fixture in your drawing and pressing the Backspace key will return the fixture to the Flight Case. The fixture will retain the position assignment.

To insert a hang structure from the Flight Case

If you drag a position from the Flight Case, you will create a pipe.

1. Expand the *Positions* branch to see all the hang structures.
2. Click and hold on the position you want to insert.
3. Drag the pipe onto the drawing.
4. Release the mouse button.
5. Use the pop up dialog box to answer the questions about the trim height and length of the pipe or use the *Interactive* button to draw the pipe in interactive mode.

   Result: If you do not choose interactive mode, the pipe will be inserted centered on the origin at the trim height specified.

   Note: Multiple positions can have the same name.

   For more information about inserting pipes please refer to "Drawing pipes" on page 6.4-9.

To insert a focus position from the Flight Case

1. Expand the *Focus Positions* branch in the flight case to see all of the Focus positions that have not yet been placed in the drawing.
2. Click and hold on the focus position you want to insert into the drawing.
3. Drag the Focus position onto the drawing.

4 Release the mouse button to place the focus position.

Flight Case Properties

The Flight Case properties affect which fixture attributes are shown.

To modify the Flight Case properties

1 From the **Options** menu, choose **View Options**.

Tip: You can also use the *View Options* tool on the *Standard* toolbar.

2 Select the fixture attributes that you want displayed in the Flight Case.

Note: The *Unit Number* is displayed by default, however, you can select all the attributes if you want.

6.11 Rendering

At any time in the CAD mode, you can generate a rendering of your set. Note that renderings in CAD mode are primarily intended for rendering your set with environmental lighting but without theatrical lighting. However, if there are any fixtures selected on your plot, they will be displayed in your rendering.

To render a lighting look

1 Display the lighting look you wish to render in the working area of the screen.

2 From the **Design** menu, choose **Render**.

Tip: You can also click the *Render Wizard* tool in the *Design* toolbar.

Result: The Render Wizard will open and guide you through a series of options. Click *Next* and *Back* to navigate through the Render Wizard.

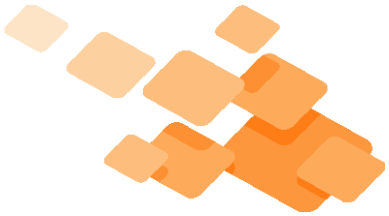
3 Click *Finish*.

Result: WYSIWYG generates the rendering of your lighting look or cue. If the rendering is saved internally, it is available as an image in Presentation mode.

For more information on rendering, refer to “Rendering” on page 8.6-1.

Chapter 7.0

Data Mode



Data mode is where you can view and edit the data for all your fixtures. Data mode operates like many other spreadsheet programs, so many of the concepts will be familiar to those who have used a spreadsheet program.

In this chapter

7.1 Layout Tabs 7.1-1

7.2 Spreadsheets 7.2-1

7.3 Patch 7.3-1

7.4 Errors 7.4-1

7.1 Layout Tabs

Beneath the work area in each mode is a series of layout tabs. These layouts provide various configurations of the views you are working with. To change layouts, click the tab that corresponds to the layout you wish to use. The Data mode contains the following layouts:

- Spreadsheet - work area contains a spreadsheet of data currently in the show database.
- H Select - work area is split horizontally, with a wireframe view above and a spreadsheet view below.
- V Select - work area is split vertically, with a wireframe view on the left and a spreadsheet view on the right.
- Patch - work area is split, with a wireframe view above and the patch window below.
- Errors - work area contains a spreadsheet displaying fixtures with errors.

Note: The wireframe views in Data mode are used for fixture selection only. Any selections made in the spreadsheet are visible on the CAD drawing and vice versa. However, to make any drawing modifications, you must return to CAD mode.

7.2 Spreadsheets

The Data mode displays your fixture information in spreadsheet format. WYSIWYG compiles a sleuth of spreadsheets which are accessible on the columns shortcut bar. All of these spreadsheets contain the same information but are sorted and filtered differently. Each column in the spreadsheet represents one fixture attribute. The following information is compiled.

Data fields

Note: Data fields identified with an asterisk are non- editable fields.

Spot - This is an assigned identifier number usually used for automated fixtures. A spot number is required for automated fixtures when using AutoFocus.

Channel - This is the assigned control channel number you will use at your control console to control the fixture. For moving lights, the channel number recorded in WYSIWYG is the starting channel number.

Position - This is the hanging position for the fixture. Positions must be entered in the Position Manager.

Unit - The unit number identifies the fixture's location on its respective hanging position.

Type - This is the fixture name.

Lens - This is the lens type.

***Hookup** - This identifies the component of a multi-circuit fixture or other device such as a scroller (ex. intensity, control).

Purpose - The purpose is a custom note that is most commonly used to describe how this fixture is being used in your show. For example, "SL Side", "Diagonal Backs". Purpose is an attribute of the fixture. It is not possible to assign multiple purposes for multi-circuit fixtures.

Colour - This is the assigned gel colour number or scroller identification.

Gobo - This is the assigned gobo number.

Focus - This is the fixture's focus position.

Patch - This is the fixture's assigned DMX channel number. This field is mandatory for simulation activity in Live mode. One show can have multiple DMX universes. A patch entry must be notated *universe.#*, where *universe* is a letter, number, or other label identifying the universe or output and # is the DMX channel number. For example, "A.1" or "Dim.26".

Circuit Name - This is an identifier note for the circuit box or multi-cable.

Circuit Number - This is the assigned circuit or multi-cable tail number.

Dimmer - This is the assigned dimmer number.

***Offset** - This field identifies the fixture's location on the hanging structure. It is a distance measurement referencing the pipe's end or center point or another point as specified.

Pan - A focus attribute measured in degrees, defining the positioning of the fixtures yoke.

Tilt - A focus attribute measured in degrees, defining the positioning of the fixture within the yoke.

Spin - A focus attribute measured in degrees, defining the fixture's yoke positioning in relation to the hang structure where 0 is down-hung, 90 is side-hung, and 180 is over-hung for example.

***Weight** - This is the fixture's weight. A fixture's weight can only be modified through the Library Browser.

Lamp Type - This is the lamp type.

Notes - This is a custom notes field.

***Footnotes** - This feature is currently disabled.

***# of Data Channels** - This is the total number of DMX channels required by the fixture.

***# of Colour Frames** - This is the number of colour frame slots that the fixture has.

***# of Lamps** - This is the number of lamps required by the fixture.

***Circuit Type** - This describes what type of device the unit should be plugged into, for example, regular dimmer, scroller power supply.

***Wattage** - This is the wattage in watts of the lamp.

***Model** - This is the fixture type.

***Cost** - This is the fixture's cost or rental cost. This field is used to estimate a show budget. A fixture's cost can only be modified through the Library Browser.

***Status** - This is the fixture's status relative to your drawing. If the fixture is HUNG it is in your drawing. If a fixture is UNHUNG, it is in the flight case. In fixture count reports, all fixtures are counted regardless of their status (unless a filter is applied).

***Console** - This identifies which console is controlling the fixture. This field references the binding settings in the device manager in Live mode.

Layer - This field indicates which layer the fixture is drawn on.

***Tag** - This is an internal code used for importing and exporting data to/from third party programs.

***Owner** - This feature is currently disabled.

Working in the spreadsheet

Data may be entered in a number of ways within the WYSIWYG file. The plot can be created and then edited, or the data may be entered in a spreadsheet and then placed on the plot. Any entries or changes are reciprocated throughout the file; changes made in Data mode will be updated in CAD mode and visa versa.

To add or modify data

Enter the desired value in the appropriate cell.

Tip: You can add or modify chunks of information at the same time by selecting a series of cells and typing.

If you are entering sequential numerical values for a field such as patch, you can use incremental data entry to facilitate your work. WYSIWYG will calculate the next available value based on the number of required channels for the previous fixture.

To input sequential numerical data

1. Enter the first value in the appropriate cell.
2. Press and hold ALT and press the down-arrow for each incremental entry you want to make.

To choose a new value

1. Select the appropriate cell.
2. Right click and choose **Pick value from list**.

   Result: This action opens a dialog with a pull down menu listing all of the values currently used in your file for you to choose from.
3. To make a new selection, click *Library* to open the appropriate library.

   Note: This feature is only available in certain fields.

   Tip: This feature is especially useful for fields such as colour, gobo, or other parameters that reference library items for which you don't necessarily know the exact value by heart.

Find and Replace Tool

The find and replace tools are used for locating and modifying data, and are column specific. The tools only search for data in the column where the tool was activated and not throughout the entire spreadsheet.

To use the find tool

1. Select a cell in the column for which you want to use the *Find* tool.
2. From the **Edit** menu, choose **Find**.

   Tip: You can also use the *Find* tool on the *Data* toolbar.
3. Type your search criteria.
4. Select the *Match whole word only* check box to search for data matching exactly what you have typed as opposed to data containing parts or all of what you have typed.
5. Select the *Match case* check box to enable a case sensitive search where uppercase and lowercase letters will affect the results.
6. Click *Find Next*.

   Result: Cells that match your search criteria are highlighted with the first match active.

To use the replace tool

1. Select a cell in the column for which you want to use the *Find* and *Replace* tool.
2. From the **Edit** menu, choose **Replace**.

   Tip: You can also use the *Replace* tool on the *Data* toolbar.
3. Type your search criteria. Enter a value to find and a value to replace it with.
4. Select the *Match whole word only* check box to search for data matching exactly what you have typed as opposed to data containing parts or all of what you have typed.
5. Select the *Match case* check box to enable a case sensitive search where uppercase and lowercase letters will affect the results.
6. Click *Find Next*, *Replace*, or *Replace All*.

   Result: If you click *Find Next*, cells that match your search criteria are highlighted with the first match active.

   Result: If you click *Replace*, cells that match your search criteria are highlighted and the first match is replaced with the value specified in step 3. You can continue clicking *Replace* to replace each match in turn.

   Result: If you click *Replace All*, all matches are replaced with the value specified in step 3 and no cells are highlighted.

Inserting fixtures in Data mode

Any fixtures inserted in Data mode will be assigned the status "unhung" and will be placed in the Flight Case. The Flight Case allows you to drag and drop "unhung" fixtures onto your drawing. For more information about the Flight Case, refer to "The Flight Case" on page 6.10-1.

To insert fixtures in Data mode

1. Click the *Fixture* tool on the *Data* toolbar.
2. Navigate to the desired fixture.
3. In the *Multiples* box, enter the number of fixtures of that type required.
4. Click *Insert*.

   Result: The fixtures are inserted below the last entry in the spreadsheet.

   Tip: If you have a shortcut created for the desired fixture, you can right click on the shortcut and choose **Insert** or **Insert Multiple**.

Inserting focus positions in Data mode

If you type a new value into the focus field of a fixture, you are essentially creating a focus position. That focus position is now available in the Flight Case for you to drag and drop into your drawing.

For more information on the Flight Case, refer to "The Flight Case" on page 6.10-1.

Inserting positions in Data mode

If you type a new value into the position field of a fixture, the *Pick a value from the list* dialog box is automatically displayed. This is because a position cannot exist in WYSIWYG unless it is recorded in the *Position Manager.* You can select from the list of positions that already exist or you can click *Manager* to open the *Position Manager* to create a new position.

If you are making this change to a fixture that was previously hung on a different position, that fixture will by unhung and sent to the *Flight Case* under its new position field. From there you can drag it out back onto the drawing.

If the position does not yet exist in the drawing, you must draw a hang structure and assign it the appropriate position name before you will be able to hang the fixture again. For more information on drawing hang structures, refer to "Hang Structures" on page 6.7-1

For more information on drawing items from the Flight Case, refer to "Entering and modifying objects in the Flight Case" on page 6.10-1 .

Inserting colour in Data mode

To insert colour in Data mode

1. Click in the colour field of the fixture for which you want to assign colour.
2. Right-click and choose **Pick value from the list**.
3. Use the pull-down menu to select a colour among those you have already used in your plot or click *Library* to open the colour library.

   Tip: You can bypass steps 2-3 and type in the value if you already now the colour you want to assign (ie. R54, L112, etc.)

WYSIWYG accomodates multiple colour entries for one fixture as follows:

- "Colour1 & Colour2" yields two colour frames with one gel in each.
- "Colour1 + Colour2" yeilds one colour frame with two gels in it.
- "Colour1 / Colour2" yields one colour frame with one split gel in it.

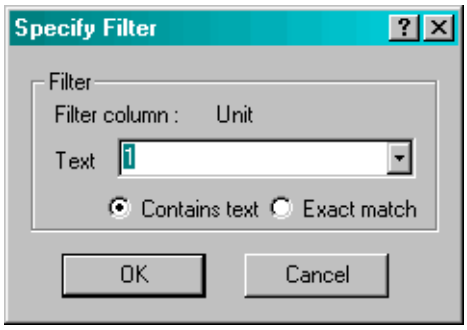
Data filters

Data filters are used for locating or isolating selective blocks of information. When a data filter is applied to a spreadsheet, only the units that met the filter criteria are displayed. Filters are applied to columns of data.

To apply a data filter

1. In the spreadsheet, right-click on the column heading you want to filter.
2. Choose **Add Data Filter.**

   Result: A dialog box is displayed, showing the column name and the available filters.



3 Select the value you want to filter. You can also type in a value and choose whether you want to filter that specific text, or filter for entries that contain that text.

Result: The spreadsheet will refresh, displaying only fixtures that meet the filter criteria.

To remove data filters

1 Right-click on a column heading.

2 Choose **Remove Data Filter.**

Result: The spreadsheet returns to its unfiltered state.

Customizing spreadsheets

There are different ways to sort and view your data. First, consider the different spreadsheets available on the *Columns* shortcut bar.

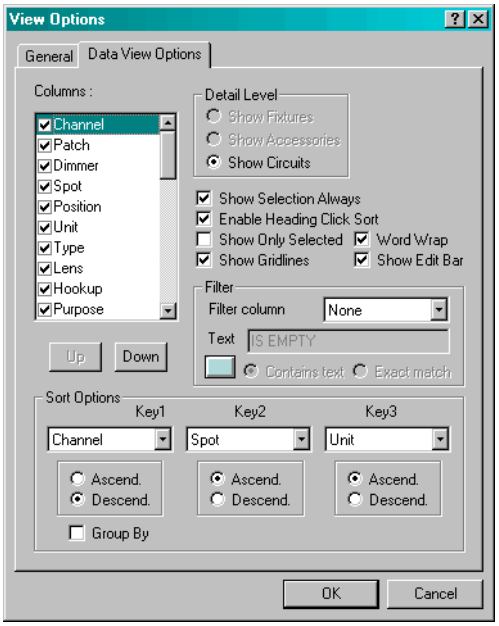
Alternatively you can modify a spreadsheet to suit your needs. Customizing a view allows you to change how the data is displayed and sorted.

To modify a data sheet

1 From the **Options** menu, choose **View Options**.

2 You can also click the *View Options* tool on the *Standard* toolbar.

3 Click the *Data View Options* tab.



4 To adjust a column's location in the spreadsheet, highlight the appropriate column heading in the *Columns* box, then click the Up or *Down* button to relocate it in the data view.

5 To hide a column in your view, deselect it by clicking on the box so that the check mark disappears.

6 To specify how entries should be sorted, choose the desired column headings in the *Key 1*, K*ey 2*, and *Key 3* drop-down lists. When fixtures have the same value in the first sort key the spreadsheet is then sorted by the values of the second sort key.

7 Click *Ascend* or *Descend* to sort the criteria in ascending or descending order, respectively.

8 Select the *Show Selection Always* check box to ensure that any fixtures selected in a plot view will automatically be selected in a data view.

9 Select the *Enable Heading Click Sort* check box to enable the ability to sort the spreadsheet by clicking on the column headings while working in the spreadsheet.

10 The *Show Only Selected* option will filter the spreadsheet to display only the fixtures that are selected in the Wireframe view.

11 Select the *Show Gridlines* check box to display the columns in tabular format, with lines separating the data.

12 Select the *Word Wrap check box to* view all the text on a line in the spreadsheet.

13 Select the *Show Edit Bar* check box to display the edit bar at the top of the spreadsheet.

14 The *Filter* section has the same functionality as the data filters discussed in "Data filters" on page 7.2-5.

15 Select the *Group By* check box to divide the spreadsheet into sections/ pages, one section/page for each value in the Key 1 field.

Creating a new spreadsheet

To create a new sheet

1 Click the *Columns* tab in the shortcut bar.

2 Right-click in the shortcut bar and choose **New Spreadsheet**.

3 Name the new spreadsheet and click *OK*.

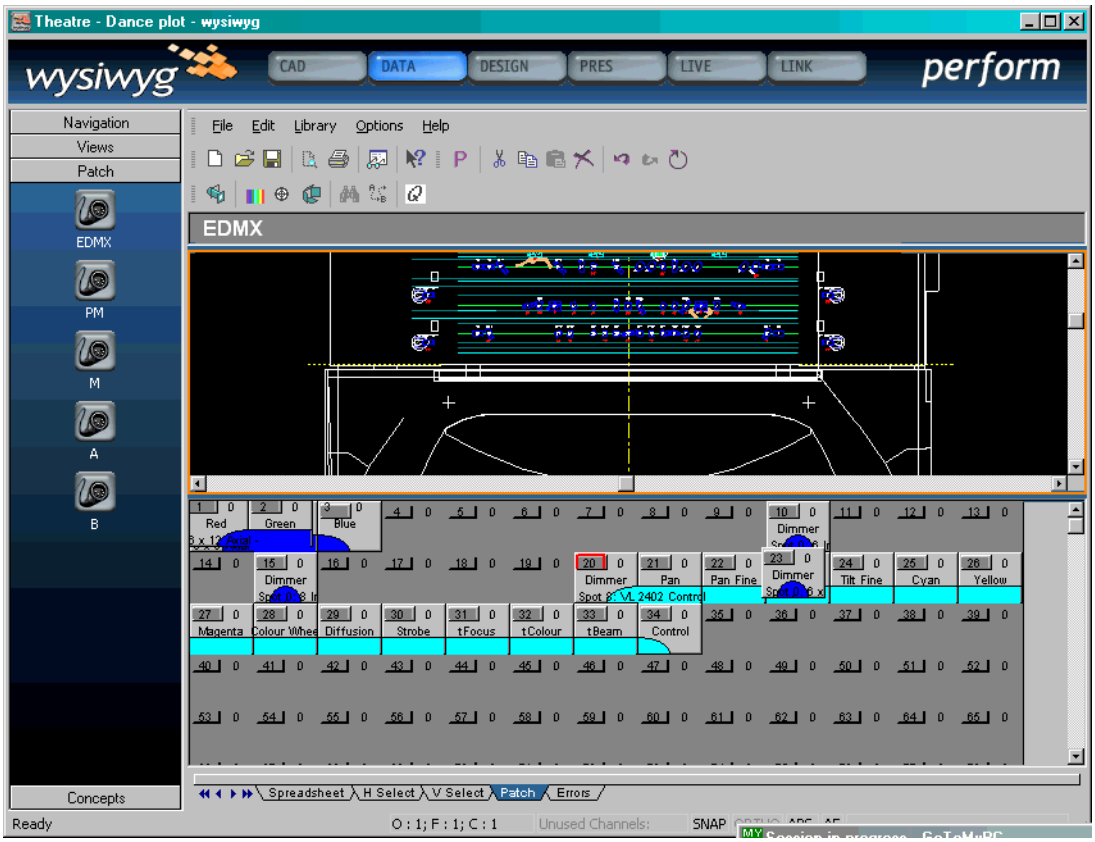
4 Scroll to the bottom of the list of *Column* shortcuts. Your new spreadsheet name should be at the bottom of the list. Click on the shortcut to view your spreadsheet.

Note: It may be easier to clone an existing spreadsheet and modify it than to start a new one from scratch. See "To clone a shortcut" on page 4.2-3 for more details.

7.3 Patch

The Patch view is a graphical representation of the patch field. It is used for two purposes:

- to create and/or edit a patch for the lights in your drawing
- to monitor incoming DMX levels when using WYSIWYG Perform for simulation.



Working in the patch view

You can view one patch universe at a time. As you add new patch universes in either a spreadsheet view or a CAD drawing (using Quick Tools), their shortcuts are created automatically. You can also create new patch universes directly from the shortcut bar.

To create a new patch universe

1 Click the *Patch* layout tab.

2 Click the *Patch* shortcut bar.

3 Right-click in the patch shortcut area.

4 Choose **New Patch**.

Result: A dialog box is displayed, prompting for the name of the new universe. Patch universes can be named anything. For example, ”A”, “Dimmers”, “Dim”, “Movers”, “Scrollers”, etc.

5 To change, delete or modify a patch universe, right-click the shortcut icon for the universe you want to change. Click on the command you want to perform.

6 To display the contents of a patch universe, click on the appropriate shortcut icon. The name of the universe will be displayed at the top left corner of the Wireframe view.

To patch fixtures in the patch view

1 Click and hold the desired fixture in the Wireframe view.

2 Drag the fixture to the desired patch location.

3 For re-patching, repeat the above steps, or click and drag the fixture patch information to the new location.

To unpatch fixtures in the patch view

1 Select the fixture by clicking on the first channel (start cell) for that fixture. It should become highlighted.

2 Right click and choose **Unpatch**.

Result: The fixture is unpatched; the value in the patch field for that fixture is deleted.

Reading the patch

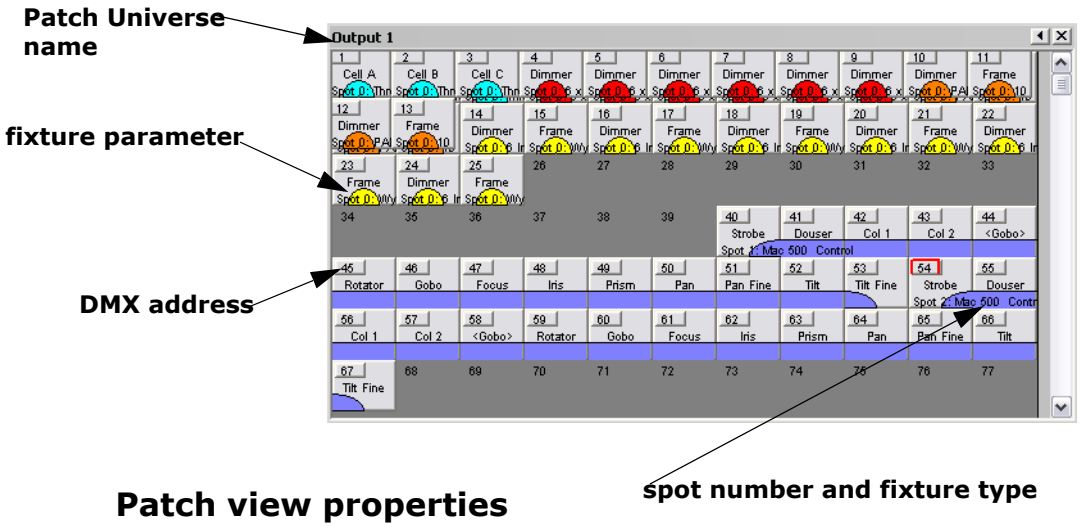
Each box in the patch universe represents one DMX channel. Fixtures are represented by colored bumps or bars that span the number of required DMX channels. The color of the bar is the same as the fixture’s CAD layer.

The number in the top left corner is the DMX address.

The spot number and fixture type are identified across the bottom of the box(es).

The fixture parameter controlled by each channel is identified in the center of the box.

If you are connected to a console and are receiving DMX signal, the incoming DMX levels are displayed in the top right corner of each box.



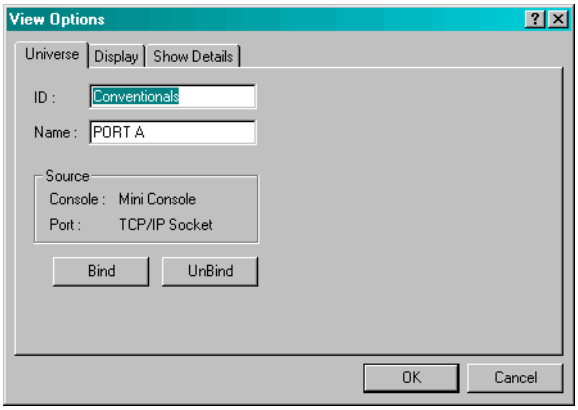
Patch view properties

Patch universe properties that can be viewed and modified include the name, display details, and tooltip information that is displayed in a patch universe.

To view the properties of a patch universe

1 Right-click on the patch icon and select **Properties**.

Result: The *View Options* dialog box is displayed.



1 Universe

Options on the *Universe* tab affect the binding/unbinding of the patch universe to a specific console or device output.

a. *ID*: the name of the patch universe shortcut.

b. *Name*: the full name of the patch universe or port ID.

c. *Source*: the lighting control console or device and port to which the patch universe is bound.

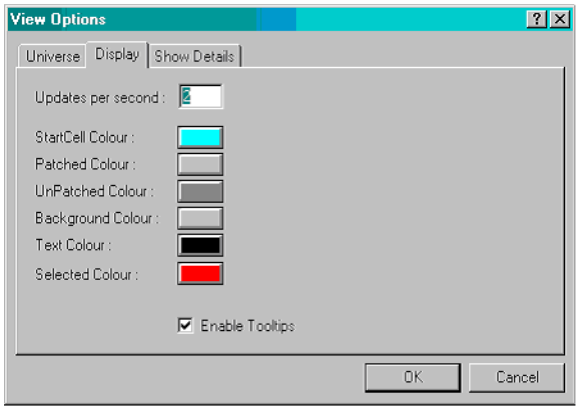
d. *Bind*: associates the patch universe to a console or device output. This applies only when a console or device has been inserted in the Device Manager.

e. *Unbind*: removes the association of the patch universe to the specified source.

Tip: You can open the *Device Manager* without having to change to Live mode by double clicking on the *AF* shortcut on the status bar.

2 Display

Options on the *Display* tab set properties such as the colour of a patch universe.



a. *Updates per second:* the frequency of updates from the console to the patch universe.

b. *StartCell Colour*: the colour of the first channel of a fixture.

c. *Patched Colour*: the colour of a cell in a patch universe that is patched.

d. *UnPatched Colour:* the colour of a cell in the patch universe that is not patched.

e. *Background Colour:* the colour of the background of the patch view.

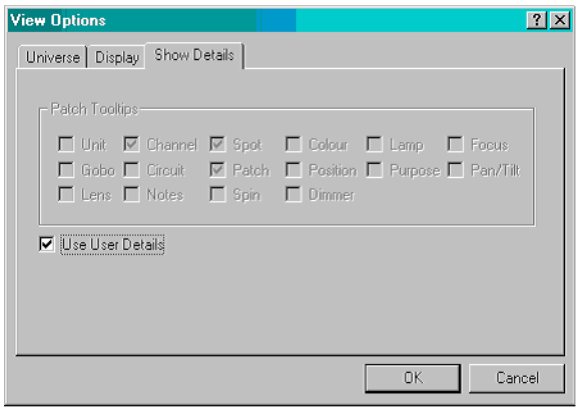
f. *Text Colour*: the colour of the text that is displayed in the patch universe.

g. *Selected Colour*: the highlighted colour of a selected fixture in the patch universe.

h. *Enable Tooltips*: select this check box to display information about the patch and the fixture when you hover over the start cell.

Show Details

Options on the *Show Details* tab define the information to be shown in the tooltips when tooltips is enabled.



a. *Use User Details:* select this check box if you want to reference the settings in *User Options* (under the *Options* menu). Click to clear the check box and specify the tooltips that you want displayed.

7.4 Errors

WYSIWYG sports an “on the fly” error checking engine. This means that as you are putting your lighting design together, WYSIWYG is continually checking for data errors such as multiple dimmers in one circuit, multiple channels in one dimmer, and various missing information.

At any time, you can view the error reports which are available on the *Errors* shortcut bar. Error reports that have errors displayed will have an "!" after their name. You can edit/fix those errors on the spot; editing data on the error sheets works the same way as the data spreadsheets explained in the previous section. As soon as you fix an error in an error spreadsheet, the entries will grey out to show that it has been fixed.

An indicator on the status bar signals you when your document contains errors. This feature can be turned on or off. WYSIWYG is also set up to trigger an error notification when errors are detected. Error notifications can be turned on or off.

Errors

WYSIWYG checks for the following errors:

Error	Definition
Dimmers > Circuit	Fixtures assigned to the same circuit have been assigned different dimmer numbers.
Patches > Circuit	Fixtures assigned to the same circuit have been assigned different patch addresses.
Channels > Circuit	Fixtures assigned to the same circuit have been assigned different channel numbers.
Patch > Dimmer	Fixtures assigned to the same dimmer have been assigned different patch addresses.
Channels > Dimmer	Fixtures assigned to the same dimmer have been assigned different channel numbers.
Channels > Patch	Fixtures assigned to the same patch address have been assigned different channel numbers.

Error	Definition
Spots > Patch	Fixtures assigned to the same patch address have been assigned different spot numbers.
Missing Position	The position field is empty.
Missing Unit	The unit field is empty.
Missing Circuit Name	The circuit name field is empty.
Missing Circuit #	The circuit number field is empty.
Missing Dimmer	The dimmer field is empty.
Missing Patch	The patch field is empty.
Missing Channel	The channel field is empty.
Missing Spot	The spot field is empty.
Missing Colour	The colour field is empty.
Missing Gobo	The gobo field is empty.

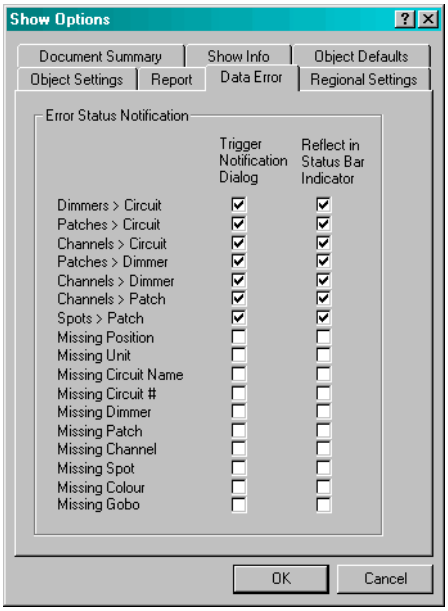
Error notifications

Error notifications occur when WYSIWYG detects an error for which an error notification trigger has been set.

To set error triggers

1 From the **Options** menu, choose **Show Options**.

2 Click the *Data Error* tab.

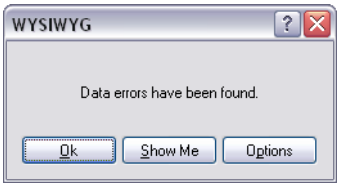


3 Select the errors you want to trigger an error notification and/or signal.

4 Click *OK*.

To manage error triggers

1 When an error notification is triggered, the following dialog box appears:



2 From this dialog box, you can perform one of the following actions:

a. Acknowledge the error and click *OK* to close the notification.

b. Acknowledge the error and click *Show Me* to open a new window that automatically navigates you to the appropriate error report.

c. Acknowledge the error and click *Options* to reset the error notification triggers as described above.

Error spreadsheet properties

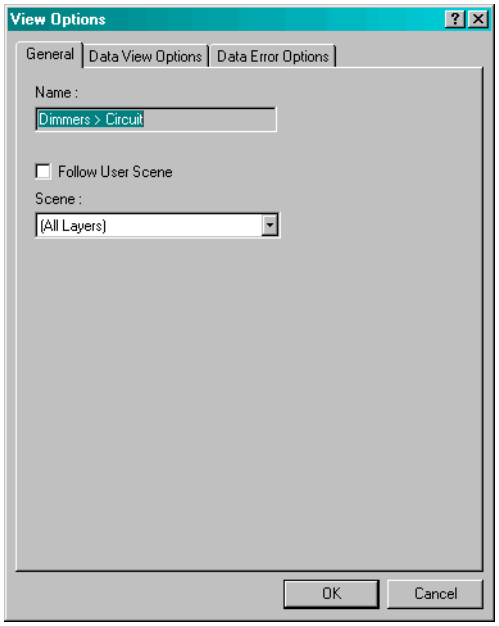
Error spreadsheets may be sorted and modified in a similar manner as other spreadsheet views.

To modify an error sheet

1 Click the *Errors* layout tab.

2 From the **Options** menu, choose **View Options**.

Result: The *View Options* dialog box is displayed.

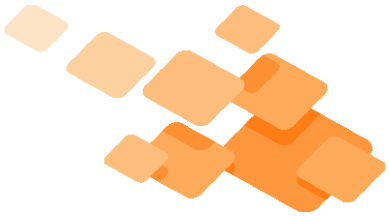


For information on the *General* and *Data View Options* tab, refer to "Customizing spreadsheets" on page 7.2-6.

On the *Data Error Options* tab, set triggers for the error displayed in the *Name* box on the *General* tab. For more information on triggers, refer to "To set error triggers" on page 7.4-2.

Chapter 8.0

Design Mode



Design mode was created to provide lighting designers with an avenue for experimentation. Use it as a troubleshooting tool or as a creative tool to help you come up with cue concepts. In Design mode, you can create static lighting looks using the design tools. Save and render those looks to output photorealistic pictures.

In this chapter

8.1 Layout Tabs 8.1-1

8.2 The Design Tools 8.2-1

8.3 Concepts 8.3-1

8.4 Palettes 8.4-1

8.5 Creating Looks 8.5-1

8.6 Rendering 8.6-1

8.1 Layout Tabs

Beneath the work area in each mode is a series of layout tabs. These layouts provide various configurations of the views you are working with. To change layouts, click the tab that corresponds to the layout you wish to use. The Design mode contains the following layout tabs:

- Wireframe - work area displays full screen wireframe view.
- Quad - work area is divided into quadrants, three of which can be modified to show plan, front, side, or isometric views. The lower-right quadrant contains a shaded view.
- Shaded - work area displays full screen shaded view.

Note: The wireframe views in Data mode are used for fixture selection only. To make any drawing modifications, you must return to CAD mode.

8.2 The Design Tools

The design tools are available in the Design and Live modes. You must have your venue, scenery and fixtures drawn in order to use the design tools; however, you do not need to have all the data entered for the fixtures in Design mode. Design mode is intended so you do not need channel numbers or a patch. The design tools allow you to create virtual lighting looks without an external control console. You can rough-in static looks and render them for design presentations.

Though the Design mode looks a lot like the CAD mode, you cannot draw anything within the Design mode. If you need to make changes to your drawing, you will need to switch over to the CAD mode.

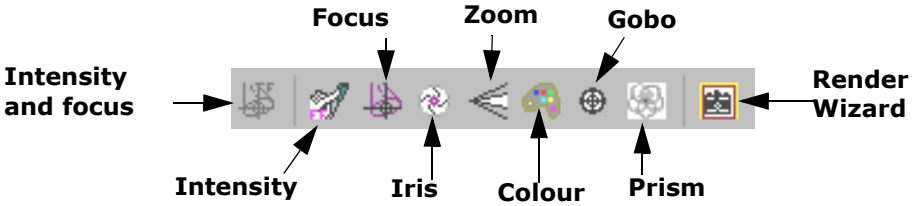
You can see the output of fixtures in the wireframe and shaded views within the Design mode. You can only select fixtures within the wireframe views, unless you use Concept shortcuts as discussed in the next section.

The design tools are individual toolbars that enable you to control certain parameters for selected fixtures. Currently there are seven design tools:

- Focus
- Colour
- Gobo
- Prism
- Intensity
- Iris
- Zoom

To turn the design tools on

1. From the **Design** menu, choose the tool name to activate or deactivate the desired design tool.
2. You can also toggle the tools on the *Design* toolbar.



Design tools have the same properties as other toolbars and can be rearranged on the screen to improve the layout of the working area. You can also resize design tools by stacking them into columns so that they take up less space on the screen.

To rearrange and resize design tools

1. Open the desired design tools using the steps above.
2. Click and drag a tool on top of another. When you release the mouse, the selected design tool will line up under the other.
3. Repeat step 2 for as many tools as will fit in a column.

4 If you overlay tools so that they are not completely visible, click on the triangle in the tools title bar. This will expand or contract the tool's window.

5 If you click and drag a tool's title bar off the column, it will expand into its own window.

Using the Design tools

Begin by selecting the desired fixtures.

To select fixtures

Select a fixture or fixtures. You can select multiple fixtures by holding the CTRL key while clicking on the fixture symbols. You can also use concepts as described in "Concepts" on page 8.3-1.

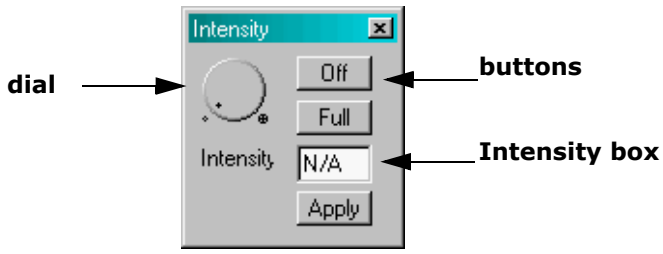
Tip: You can also right-click and drag a box around a section of your drawing to select fixtures. This displays a context-sensitive menu displaying all the fixture types contained within that box. You can then easily select "All Mac500", for example.

Result: Selected fixtures are highlighted in green.

Intensity tool

To use the intensity tool

The intensity tool provides many options for setting intensity levels.



Click the buttons for the extreme settings of full intensity (100%) and off (0%). Use the dial to achieve any of the settings in between. To use the dial, click and hold the dial and move the mouse up or down. The chosen intensity level is displayed in the *Intensity* box as you move the mouse.

Alternatively, you can enter a specific percentage by typing the number in the *Intensity* box and clicking *Apply*. If multiple fixtures are selected, the intensity level entered will be assigned to all fixtures. Selecting multiple fixtures that have varying intensity levels will cause "N/A" to be displayed in the *Intensity* box.

Zoom and iris tools

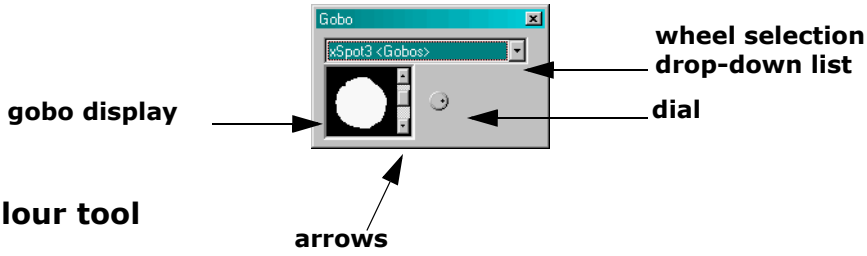
To use the zoom and iris tools

The iris and zoom design tools both work the same way. Use the buttons for the extreme settings of tight or wide. Use the dial to achieve any of the settings in between. To use the dial, click and hold the dial and move the mouse up or down.

Gobo and prism tools

To use the gobo and prism tools

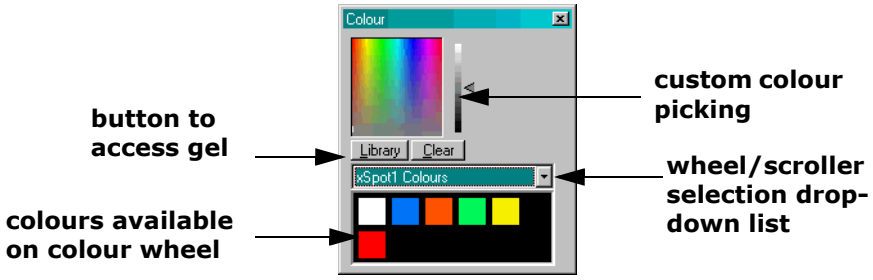
The gobo and prism tools work the same way. Specify the fixture type (if you have more than one selected), or the specific wheel to control (if that fixture type has multiple wheels) using the pull-down menu. Note that only fixtures with gobo or prism parameters will appear on the list. Use the gobo/prism wheel scroll arrows to select the desired gobo/prism from the wheel. If no gobo/prism is desired, leave the wheel in the open slot. The dial will rotate this gobo if a rotating gobo is selected.



Colour tool

To use the colour tool

The colour tool offers many options for colour selection.



Click the custom colour box and use the greyscale arrow to adjust your selection. The fixture's output will depend on its capabilities. If the fixture has CMY or colour mixing capabilities, the output will be the exact colour you selected. If the fixture is only equipped with a colour wheel or a non mixing scroller, the fixture will output the closest colour to that selected based on the available colours on the wheel.

The same principal works if you select a colour from the library by clicking the *Pick From Library* button. The wheel/scroller selection drop-down list is as discussed for the gobo wheel and with that selection all of the available colours are displayed in the order that they appear on the wheel or scroll.

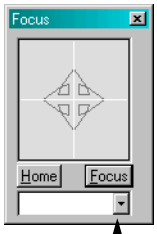
Focus tool

The focus tool works with fixtures that have variable pan and tilt parameters. Conventional fixtures must be focused in the CAD mode.

To use the focus tool

In the focus trackpad, click, hold, and drag the mouse to control the position of the light beam. The *Home* button resets the fixture to pan = 50% and tilt = 50%.

To send the fixture's beam to a specific location, click the *Focus* button, and then click in your wireframe drawing. Use the focus position drop-down list to focus the beam(s) of light toward one of the focus positions you created in the CAD mode.



focus position drop-down list

Note: The Focus tool does not take into account the current viewing position or fixture settings. In one orientation, dragging to the right on the trackpad will result in the beam moving left but rotating the fixture 180 degrees and dragging to the right should cause the beam to move to the right. This effect occurs with both moving mirror and moving head fixtures.

8.3 Concepts

Concepts are custom groups of fixtures.

Concepts can be created in CAD, Data, Design, or Live mode in any view that allows fixture selection.

Creating concepts

Creating a concept allows you to select a group of fixtures at the same time.

Concept shortcuts also enhance working with design tools in shaded views. As design tools work with the current fixture selection, opening multiple concept shortcuts allow you to change different groups of fixtures, rather than having to select them individually in the Wireframe view.

Concepts are saved on the *Concept* shortcut bar.

To create a new concept

1. Select the fixtures that you want to group. If you are unfamiliar with selecting objects, refer to "Selecting" on page 6.5-1.
2. Right-click on the Concept shortcut bar and then choose **New Concept**.
3. Enter the name of the concept and then click *OK*.

   Result: The concept is saved on the Concept shortcut bar.

Updating concepts

You can modify a Concept shortcut by adding or removing fixtures as required.

To update concepts

1. Select the set of fixtures that you want to include in the concept.
2. Right-click on the *Concept* shortcut that you want to update, and then choose **Update**.

   Result: The *Concept* shortcut is modified to include only the fixtures that you selected.

Working with concepts

When you select a Concept shortcut, any currently selected objects are deselected and the Concept selection set is selected instead.

If you press the CTRL key while selecting concepts, the concept is added to the current selection set.

If you press the CTRL and SHIFT keys while selecting concepts, the concept is removed from the current selection set.

Notes:

1. In a Concept shortcut, multi-cell fixtures such as cyc lights can be selected either by cell or fixture. If the concept was created in Design mode using cells, the entire fixture is selected in the other modes. If the concept was created using fixtures, all cells are selected when in Design mode.

2. When you replace a multi-cell fixture that is selected by circuit with a different multi-cell fixture that has a different number of circuits, the replacement fixture is selected by fixture and not by circuit.
3. When you delete a fixture, the fixture is removed from all concepts that include that fixture.
4. When you replace a fixture using the **Replace Fixtures** menu option, any concepts referring to that fixture are updated to refer to the fixture replacement.
5. Any fixture added to a document will not initially be part of any concept.

8.4 Palettes

You can create palettes of colour to facilitate colour selection. A palette is a reference to a specific colour and can be used on all colour-changing fixtures and scrollers.

Creating palettes

Creating a Palette shortcut is helpful if you want to save your favourite CMY colour mixes for later use.

You can create palettes in Design mode only.

To create a palette

1. Select the fixture from which you want to save the colour.

   Note: You can create a palette from only one fixture.

2. Right-click on the *Palette* shortcut bar and choose **New Palette**.
3. Enter the name of the palette and then click *OK*.

   Result: The CMY value is saved as a palette in the Palette shortcut bar. This value can be a CMY mix, a colour from the colour wheel or scroll, or a combination of colour sources.

Updating palettes

You can modify a Palette shortcut by modifying or replacing colours as required.

To update a palette

1. Select the fixtures from which you would like to record the palette.
2. Right-click on the *Palette* shortcut that you want to modify, and then choose **Update**.

   Result: The Palette shortcut is updated with the new CMY values.

To apply a palette

Applying a colour palette is the equivalent of setting up colour parameters for the selected fixtures, except much faster.

1. Select the fixture(s) for which you want to assign the specific colour value.
2. Click on the desired palette shortcut.

   Result: The fixtures will be assigned the recorded copy value. If the fixture has CMY or colour mixing capabilities, the output will be the exact colour you selected. If the fixture is only equipped with a colour wheel or a non mixing scroller, the fixture will output the closest colour to that selected based on the available colours on the wheel.

8.5 Creating Looks

Saving lighting looks

Changes made with the design tools update the currently selected look. The name of the currently selected look is displayed at the top of the working area. Before making changes, always check that the currently selected look is the one you want to modify.

To create a new lighting look

1. In the shortcuts bar, click the *Looks* tab.
2. Right-click in the Looks shortcut area and click **New Look**.
3. Enter the name of the new look.
4. Click *OK*.
5. Scroll to the bottom of the *Looks* shortcuts list.
6. Click on the shortcut for your new look.
7. Use the design tools to create your new lighting look.

   Tip: To build on an existing look without loosing it, clone the look shortcut and work from the copy of the shortcut. For more information on cloning shortcuts, refer to "To clone a shortcut" on page 4.2-3.

8.6 Rendering

In WYSIWYG, lighting looks can be rendered to produce high quality images of your set and lighting. Rendering uses a full range of effects including depth of field, motion blur, soft shadows, and antialising settings to produce a photorealistic image. An easy-to-use Render Wizard provides interactive editing and advanced visual effects.

To render a lighting look

1. Display the lighting look you wish to render in the working area of the screen.
2. From the **Design** menu, choose **Render**.

   Tip: You can also click the *Render Wizard* tool in the *Design* toolbar.

   Result: The Render Wizard will open and guide you through a series of options. Click *Next* and *Back* to navigate through the Render Wizard.
3. Click *Finish*.

   Result: WYSIWYG generates the rendering of your lighting look or cue. If the rendering is saved internally, it is available as an image in Presentation mode.

Each of the steps asks you to set a number of related options and from there you can go on to the next page or click *Finish* to complete the rendering. If you click *Back* to change an option in a previous step, you can click *Finish* without having to go through each of the steps again.

During any step (up to step 6), you can click on the image and change the perspective (using the arrow, CTRL and PAGE UP/PAGE DOWN keys or the mouse). The Render Wizard calculates the rendering based on the state of the image composition and light settings at the moment you click *Finish*.

Saving render settings

The options that you set in the Render Wizard can be saved as a shortcut for quick access to the settings at a future time. There are two ways of saving a Render shortcut:

- from the Render Wizard
- from the Render shortcut bar, in a similar manner as other shortcuts

The Render shortcut bar is prepopulated with two shortcuts: Quick Render and Full Render. Quick Render shortcut provides settings for a small lower-quality render, while the Full Render shortcut provides settings for a larger high-quality rendering.

To create a shortcut from the Render Wizard

1. From the **Design** menu, choose **Render**.
2. While working through the steps in the Render Wizard you can, at any time, click the *Add Shortcut* button to save the current settings as a shortcut.

3 Enter a name for the shortcut.

4 Click *OK*.

Result: The shortcut is added to the Render shortcut bar.

To create a shortcut from the Render shortcut bar

1 Right click in open space on the Render shortcut bar and choose **New Render**.

2 Enter a name for the new render.

3 Click *OK*.

Result: The Render shortcut is saved on the Render shortcut bar. The shortcut is preconfigured with the default values for each of the eight steps. These defaults are the same values that are used when you choose **Render** from the menu bar or the toolbar.

To modify a Render shortcut

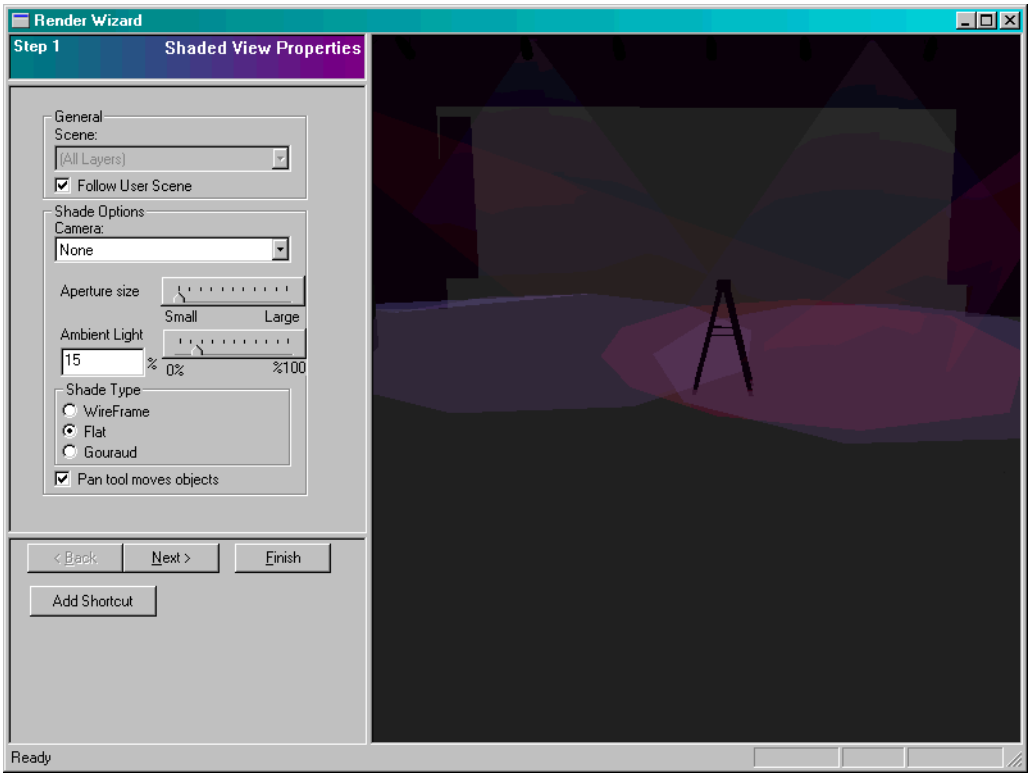
1 Double-click on the Render shortcut that you want to modify.

Result: The Render Wizard opens, with the values that are saved in the shortcut.

2 Modify the options as required.

Result: The settings are recorded in the shortcut and are available the next time the shortcut is selected.

Step 1 - Shaded View Properties



Options in Step 1 affect the composition of your image.

a. *Scene*: select a scene for the rendering. Scenes are groupings of layers used to organize a plot. For more information on scenes, refer to "Scenes" on page 6.5-15.

b. *Camera*: select the camera for the rendering. Cameras are set up in CAD mode. For more information on cameras, refer to "Drawing cameras" on page 6.4-9.

If you select a camera, you can adjust the aperature of the camera. This adjustment varies the depth of field of the final image. The larger the aperature, the "fuzzier" the rendering. The focus point is always the target of the camera.

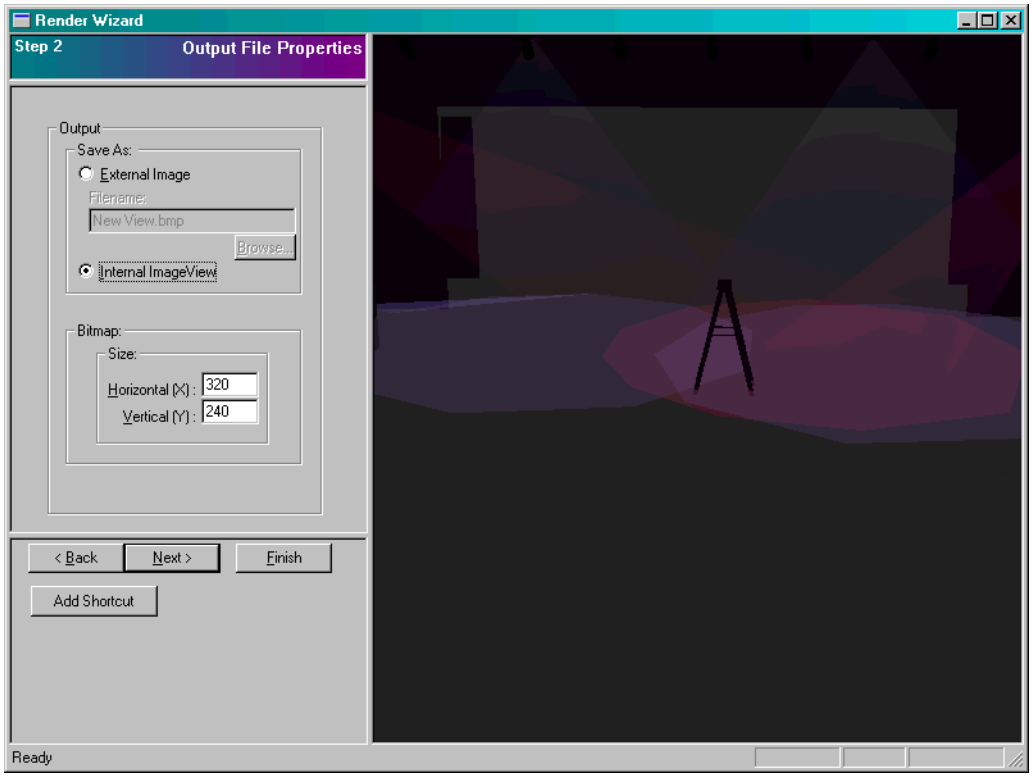
Camera target: By default, the target is visible. This is to aid in the composition of the image. It will not, however, be visible in your final rendering. To turn the target off (that is, set to not visible), right-click in the preview pane and choose *View Options*. For more information on shaded views, refer to "Modifying shaded views" on page 4.6-1.

c. *Ambient Light*: use the slider to set the ambient light level. This adjusts the overall light level where 0% is complete black and 100% is a bright room. This does not affect the intensity of the fixtures in your lighting look.

d. *Shade Type*: shade type does not affect the outcome of your rendering. Rather it allows you to set your beam quality preference as you work through the Render Wizard.

e. *Pan tool*: Pan tool does not affect the outcome of your rendering. Rather the tool rotates the model on the target (if checked) or rotates the camera around the target (if unchecked). Both settings help you to compose your image.

Step 2 - Output File Properties

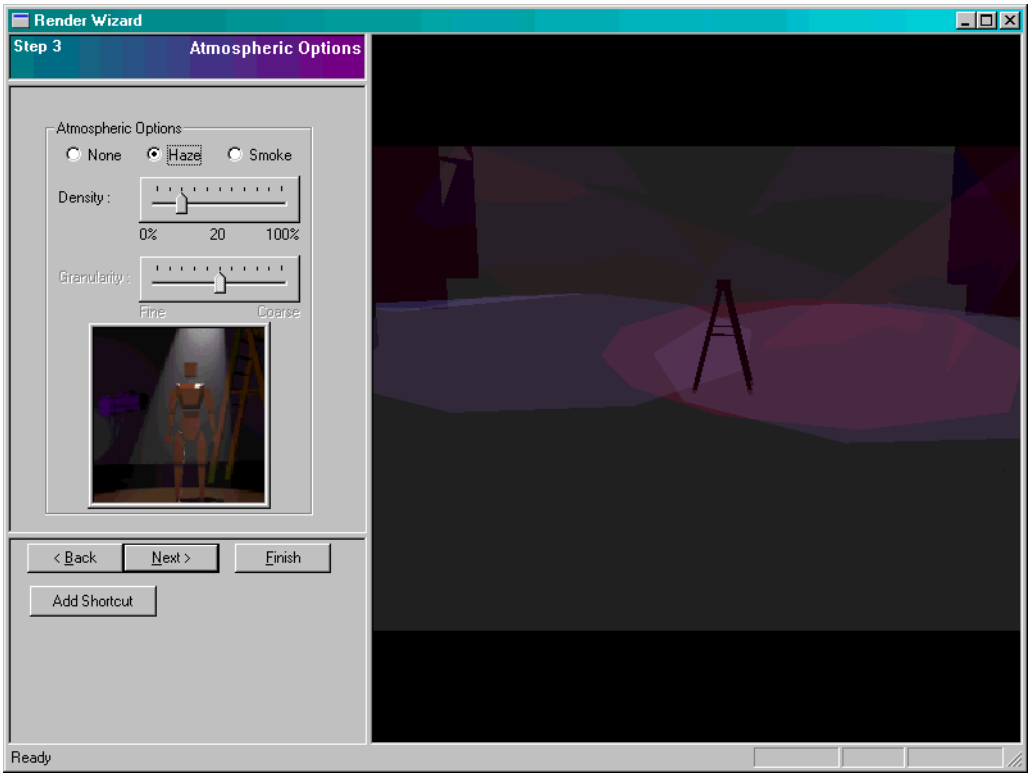


Options in Step 2 affect where your final rendering is to be stored.

a. *Save as*: saves the final rendering as an external or an internal image. If saved as an external image, the result is stored on a drive as a bitmap (.bmp), jpeg (.jpg), or Targa file. Click *Browse* to specify the file name and format. If saved as an internal image, the result is stored as an image in Presentation mode.

b. *Bitmap*: enter a value in pixels for the horizontal and vertical dimensions of the final image.

Step 3 - Atmospheric Options



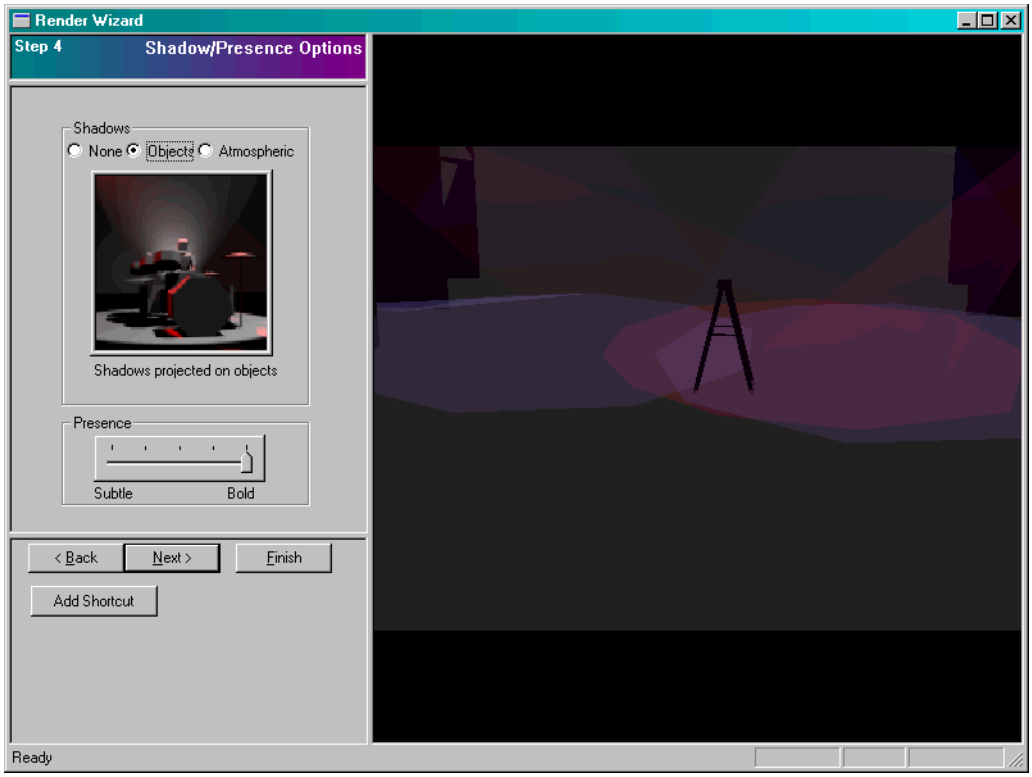
Options in Step 3 affect the atmospheric conditions in your rendering.

a. *Atmospheric Options*: the options set here will affect the atmospheric conditions that are visible in your final rendering. To simulate atmospheric dust or fog, select *Haze*. For a result of additional opacity, select *Smoke*.

b. *Density*: use the slider to set the visibility of the haze particles. A setting of 0% indicates that the particles cannot be seen.

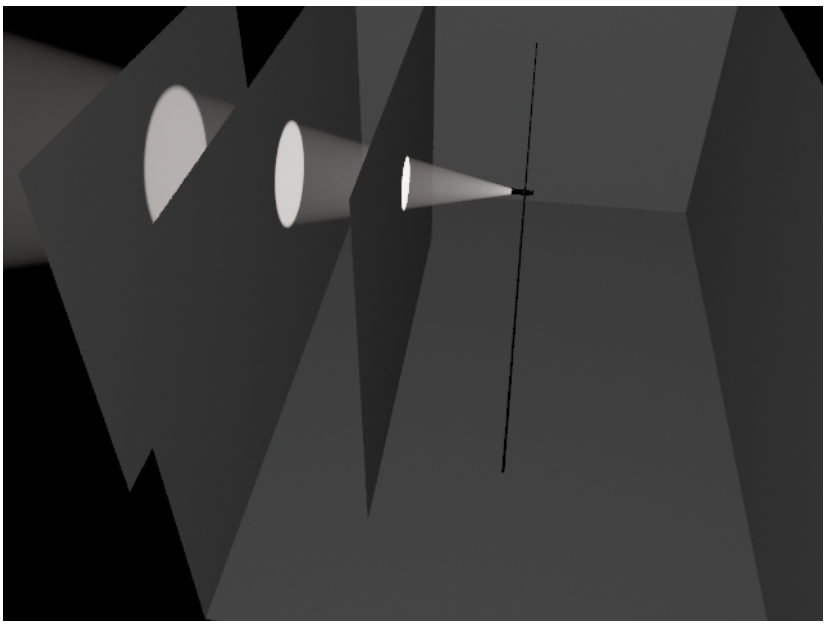
c. *Granularity*: use the slider to set the granularity of the smoke. This will affect the fineness of the smoke particles.

Step 4 - Shadow/Presence Options

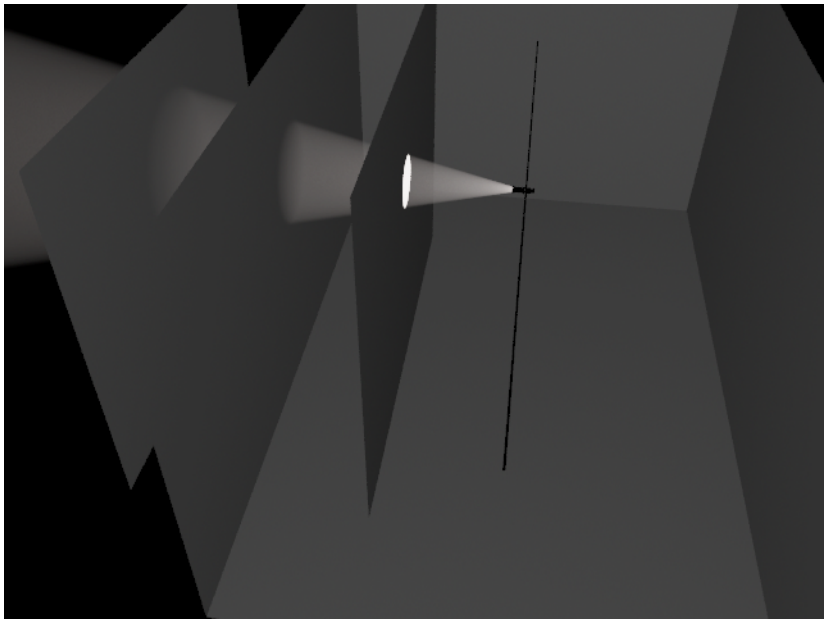


Options in Step 4 affect the projection of shadows in your rendering.

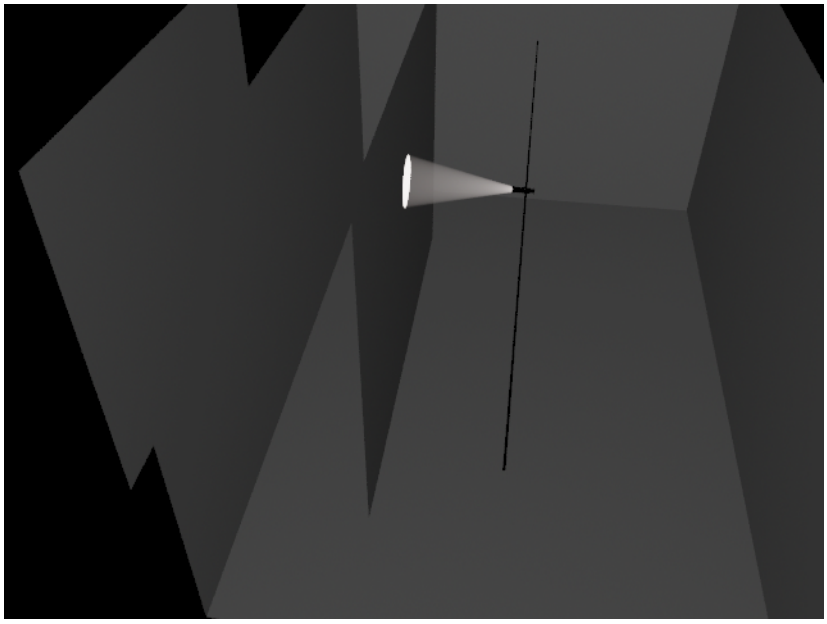
a. *Shadows*: select how you want shadows to be projected in the final rendering. Choosing *None* indicates that no shadows will be calculated by WYSIWYG; that is, objects in the path of the light beam will not create shadows. Instead, all objects in the path of the virtual beam will be illuminated, even if an object would normally block the beam from reaching another. As illustrated in the rendering below, all three surfaces are illuminated and the beam continues into infinity.



To calculate which objects the beam of light will hit and which objects it will not hit, select *Objects*. Objects in the path of the virtual beam will not be illuminated by the beam if the beam has already been blocked by another object. However, WYSIWYG still does not calculate where the beam will stop. As illustrated in the rendering below, you will see that the first surface is illuminated but that the beam is still continuing into infinity.



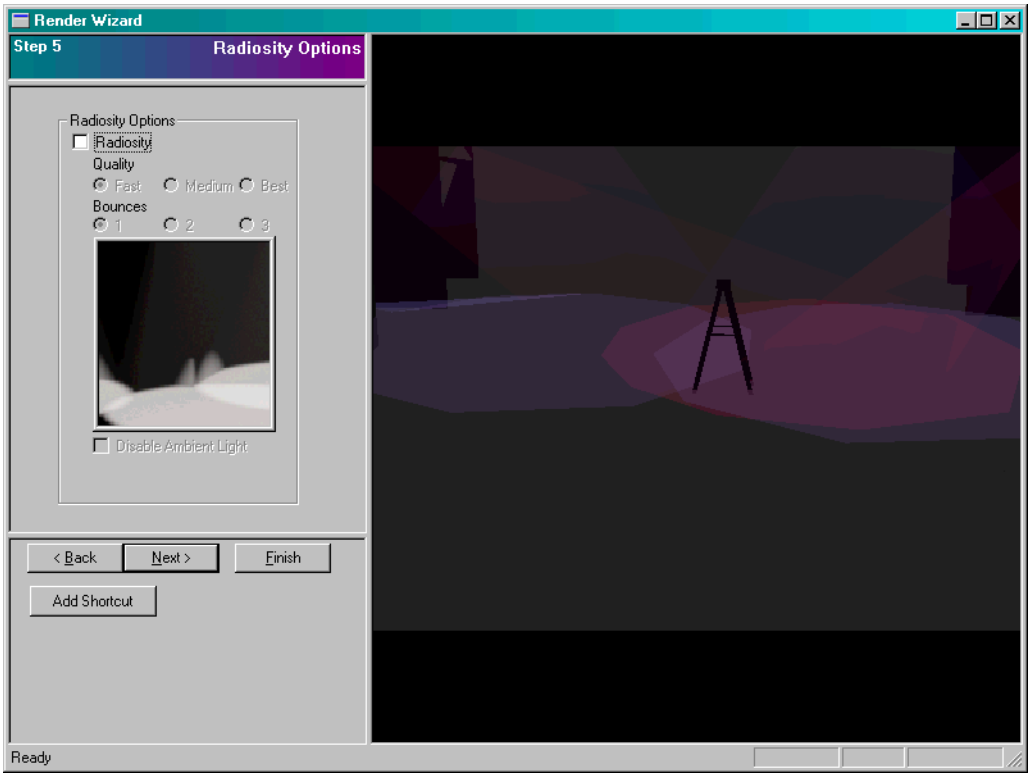
Select *Atmospheric* to calculate which objects the beam of light will hit and where the beam of light will end. This is a true representation of how light behaves. As illustrated in the rendering below, the first surface is illuminated and the beam is also stopped there.



b. *Presence*: use the Presence slider to adjust how accurately WYSIWYG calculates the effective intensity of each fixture. When set to *Subtle*, WYSIWYG accounts for gel transmission, colour temperature, fixture

efficiency, and lamp. When set to *Bold*, all fixtures are given the same effective intensity.

Step 5 - Radiosity Options

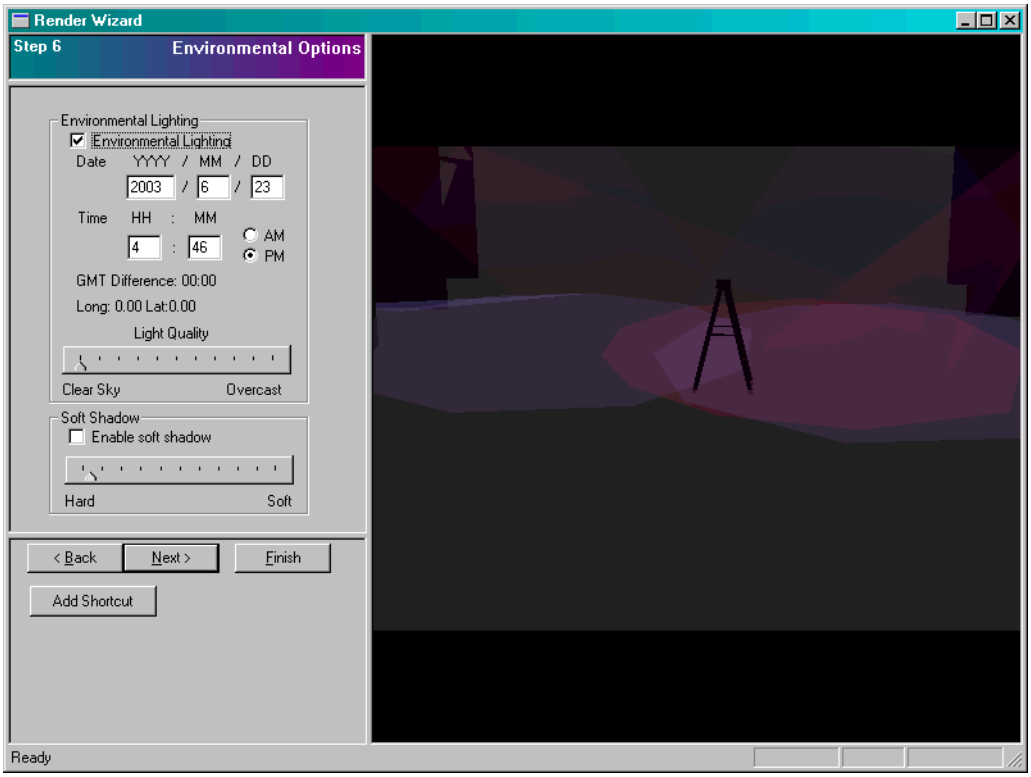


Options in Step 5 affect the radiosity level. Radiosity is defined as the combined processes of emission, transmission, and absorption of rays or reflecting beams of light. When using Radiosity, it is not necessary to use ambient lighting.

a. *Radiosity Options*: if you would like to generate a rendering that uses radiosity, select the *Radiosity* check box. *Fast* radiosity results in a rendering with constant, flat lighting. Each subsequent option shows more calculated lighting. The number of bounces determines how many levels of reflection are permitted.

b. *Disable Ambient Light*: select the *Disable Ambient Light* check box to ignore the ambient light settings from step 1.

Step 6 - Environmental Options



Options in Step 6 affect the presence and quality of environmental or outdoor lighting. Note that for environmental lighting to have an effect, your show must not be contained within a venue.

a. *Environmental Lighting*: if you would like to generate a rendering that considers outdoor lighting conditions, select the *Environmental Lighting* check box.

b. *Day* and *Time*: the day and time of the rendering determines the position of the sun and/or moon, and the resulting amount of light available. The position of the sun is also determined by the geographic settings that are configured in Show Options and the north direction that is set in CAD mode. For more information on regional settings, refer to "Regional settings" on page 5.2-10. For more information on setting the direction that is north, refer to "Drawing a compass" on page 6.4-10.

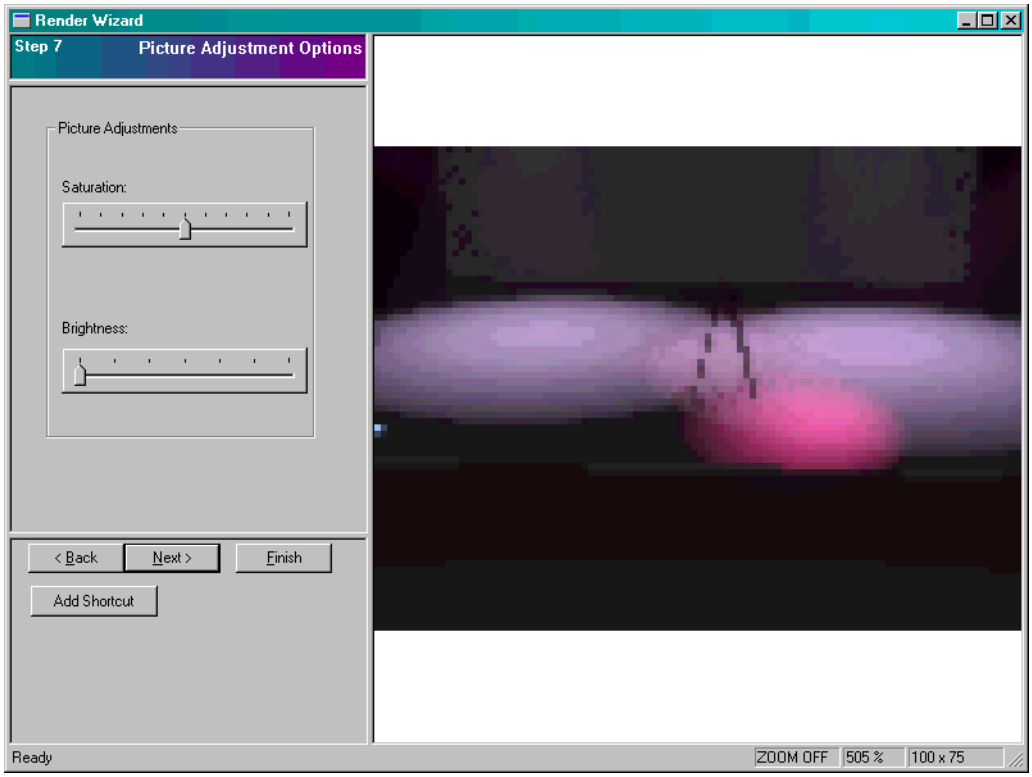
c. *Light quality*: the amount of light available determines the brightness of the resulting rendering. Use the *Light Quality* slider to adjust the quality of light.

d. *Soft Shadow*: the softness of shadows created by sunlight can be set using the *Soft Shadow* slider. The softer the shadow, the less sharp the resulting shadow.

PREVIEW

When you click *Next* in Step 6, the Render Wizard calculates a small preview of the rendering and shows it in the right-hand view. It will always zoom to fit the full size of the rendered view first, but you can click in the view and zoom in and out using the PAGE UP/PAGE DOWN keys. Use the preview to fix any glaring errors in your rendering before you continue on to the final rendering.

Step 7 - Picture Adjustment Options

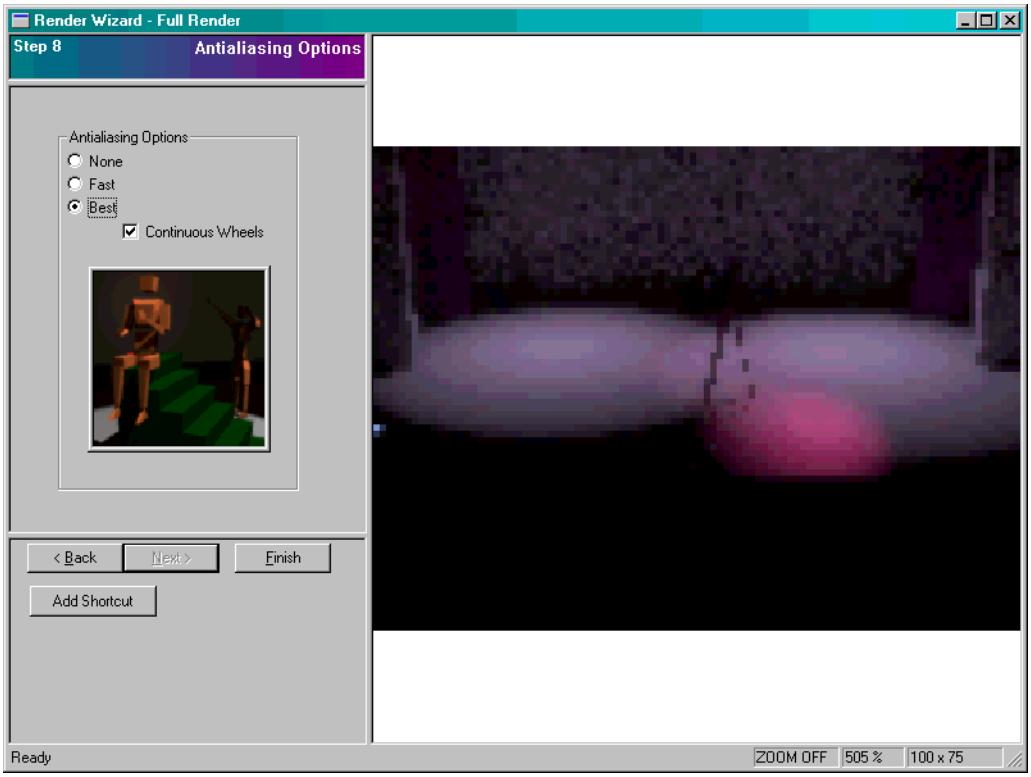


Options in Step 7 affect the quality of your final image.

a. *Saturation*: this is comparable to the exposure on a camera; it determines which pixels are represented as pure white.

b. *Brightness*: this slider adjusts all pixels equally.

Step 8- Antialiasing Options



Options in Step 8 affect the antialising settings. Antialising is a method of smoothing out and sharpening rough or jagged edges of images to produce a more polished result. This method subsamples the pixel area and averages the results of neighbouring samples to reduce the areas of high contrast (edges).

a. *Antialiasing Options*: choose how you want edges to appear in your final rendering. The *Fast* method samples the pixels quickly but does not produce the highest quality.

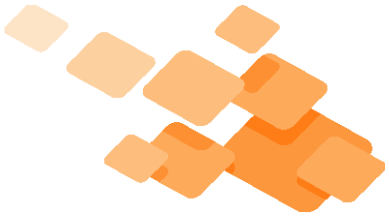
When you click *Finish*, the real rendering will take place. Depending on the options selected and the complexity of the scenery and lighting, it may take some time to compute. You may minimize the window and continue working on other parts of your document as it renders. You can use the arrow keys, PAGE UP/PAGE DOWN, mouse or scroll bars to explore the picture as it is being produced. The final image will be saved to the destination you specified in step 2. A sample of a final rendered image is shown below.

Tips

- Setup your shot accurately using a camera.
- You can always click and drag in the right-hand view to adjust your shot before you render.
- Make sure you use the appropriate aspect ratio (height vs. width) for the output device you will be presenting the pictures on. TV is 4x3 (normally 640x480), HDTV is 16x9.

Chapter 9.0

Presentation Mode



The Presentation (Pres) mode contains all of the tools necessary for creating professional printouts of your show document including reports, plots, and images.

In this chapter

9.1 Layout Tabs	9.1-1
9.2 Reports	9.2-1
9.3 Plots	9.3-1
9.4 Images	9.4-1
9.5 Worksheets	9.5-1
9.6 Pipe Tapes	9.6-1

9.1 Layout Tabs

Beneath the work area in each mode is a series of layout tabs. These layouts provide various configurations of the views you are working with. To change layouts, click the tab that corresponds to the layout you wish to use. The Pres mode contains the following layout tabs:

- Reports - work area displays reports formatted for printing.
- Plots - work area displays plots formatted for printing.
- Images - work area displays bitmap or jpeg images stored with the show file.
- Worksheet - work area contains spreadsheets editable by the user.

9.2 Reports

The Reports layout tab opens a series of pre-formatted reports generated by WYSIWYG, containing the data entered in the CAD and Data modes. These reports are available for you to use as is or modify to suit your needs. Unlike the spreadsheets in the Data mode, you cannot edit fixture data here. Reports are designed to be print-out documents. You can however modify the setup and layout of these reports.

Working with reports

To modify a report

1 From the **Options** menu, choose **View Options**.

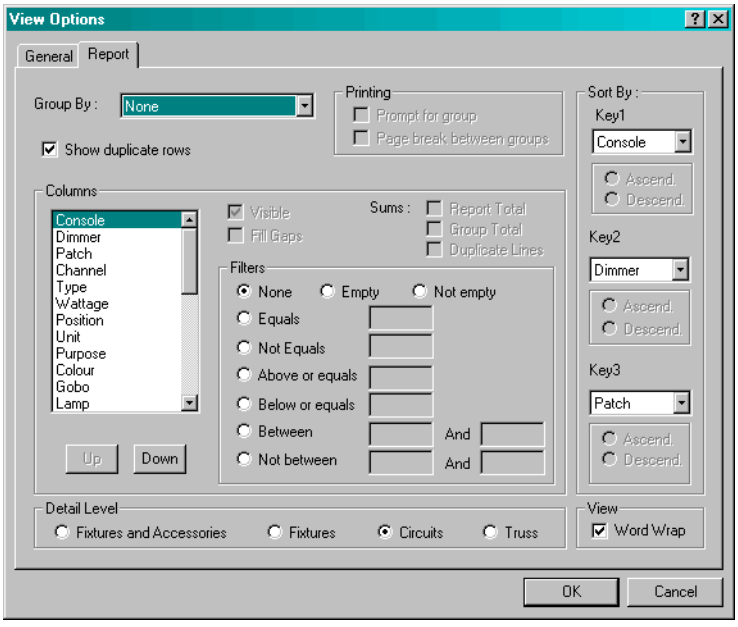
Tip: You can use the *View Options* tool on the *Standard* toolbar.

2 Click the *General* tab.

3 You can modify the following:

- The *Scene* pull-down list allows you to select a scene for the report. This will filter your show information based on the scene selected. Only fixtures on the active layers will be displayed in the report. For more information on scenes, refer to "Scenes" on page 6.5-15.

4 Click the *Report* tab.



5 You can modify the following:

- The *Group By* drop-down menu allows you to select a data field for grouping the data. These groups appear as separate tabs/pages at the bottom of a report. In the case of an Instrument Schedule, fixtures are grouped by hanging position. Each position appears on a separate page.

- The *Columns* list displays the columns available for inclusion or exclusion in a report. You can display the columns in any order by arranging the order in this list. The top of the list represents the left-hand column on the page and the bottom of the list represents the right-hand column. To move a column up or down in the list, click its name and then use the *Up* and *Down* buttons.
- There is a special column in the *Columns* list called *Count*. The *Count* column will display a count of all fixtures that meet the unique requirements of all the filters applied to all columns.
- The *Filters* group box contains settings for the column selected in the *Columns* list. The filters can be used to extract specific information from reports. For example, you can create a report that displays all fixtures that have notes. To do this, select the *Notes* column, select the Visible check box, and then select the filter *Not Empty*. You will also have to make *Position*, *Unit* and *Channel* visible so that you know which fixtures have these notes.
- Select the *Report Total* check box if you want to report the total for the report.
- Select the *Group Total* check box if you want to report totals for all groups.
- Select the *Duplicate Lines* check box if you want to report the total of all duplicate lines.
- The *Sort By* box contains the settings for how the report will be sorted. The key sorts fixtures in alphabetical or numeric order. When fixtures have the same value in the first sort key the report is then sorted by the values of the second sort key and so on for the third sort key. For example, the standard Instrument Schedule sorts fixtures by position as *Key 1*, and unit number as *Key 2*. When a column is selected as a key, the visible option will be checked and greyed out. Columns that are sort keys must be visible.
- In the *Detail Level* group box, you can specify how many rows each fixture spans. For example, if you check *Fixtures*, a three-cell cyc light will take up one row. If you check *Circuits*, that same fixture will take up three rows, one for each cell/circuit. To issue a report for truss, select *Truss* in the *Detail Level* box.
- Select the *Word Wrap* check box if you want to see all the text on a line.

6 When done, click *OK*. The report will refresh and display any changes immediately.

To create a new report

1 Click the *Reports* shortcut tab.

2 Right-click in the reports shortcut area and choose **New Report**.

3 Enter the name of your new report.

4 Click *OK*.

Result: Your report shortcut will be placed at the bottom of the list.

Tip: It is recommended that you clone an existing report and modify it rather than creating a new report.

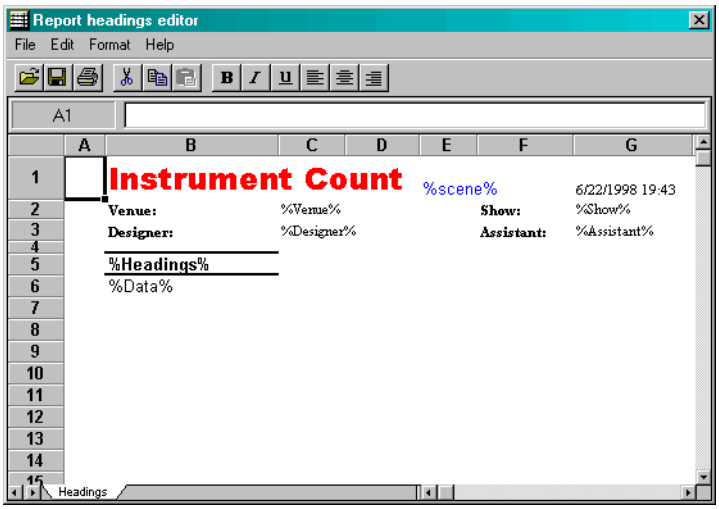
Redesigning headings

You can customize the fonts and styles used in the headings and text of your reports.

To change the heading format

1 Right-click the report and choose **Redesign Headings**.

Result: A new window opens displaying the headings for the report, with a new menu and toolbar for formatting text.



2 From the **Format** menu, use the alignment, font, border, edit and colour controls to customize the heading text.

3 Save your changes and close the window.

Result: The selected report headings are modified. You must repeat this procedure for each report whose headings you want to modify.

Although it is not possible to apply the newly designed headings to all reports at the same time, it is possible to save a heading file that can be applied to all other reports in turn.

To save a heading file

1 Redesign the headings of the selected report as described above.

2 Before closing the *Report Headings Editor*, from the **File** menu choose **Save as**.

3 Type a name for the Excel format heading file in the *File Name* field.

4 Click *Save*.

5 Close the *Report Headings Editor*.

Result: The heading file is saved in the WYSIWYG library folder unless specified otherwise.

6 You will be prompted to load the new heading filename. Click *Yes*.

Note: Since these files are saved outside of the show document. They can be applied to reports in any WYSIWYG show files.

To apply a heading file

1 Select the report for which you want to apply the heading file.

2 Right click and choose **Redesign headings**.

Result: The *Report Headings Editor* opens.

3 From the **File** menu, choose **Open**.

4 You will be prompted to save changes, select *No*.

Result: The *Load Heading File* dialog opens.

5 Browse to find the heading file you want to apply.

6 Click *Open*.

Result: The saved heading file is applied.

7 Save and close the *Report Headings Editor*.

8 You will be prompted to load the new heading filename. Click *Yes*.

Column widths

You can change the column widths in the report itself. A double-headed arrow will appear when you float the cursor between the column separations at the very top of the report. Double click the column headings to have the column size to fit (auto-fit) the contents.

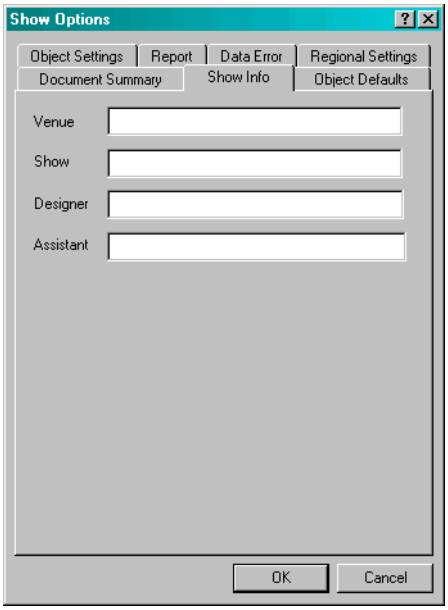
Show information

The edit fields in the show info window are venue, show, designer, and assistant. The entries in these fields appear in the headings of all reports and in the WYSIWYG default title block (for plots).

To input show info

1 From the **Options** menu, choose **Show Options**.

2 Click the *Show Info* tab.



3 Enter all the pertinent information in the boxes.

4 Click *OK*.

Result: All reports will be updated.

9.3 Plots

The Plots tab opens the plot design and storage area. Plots are created by arranging various plot, report, image, and other items on a defined paper size.

There are two default plots generated by WYSIWYG included in the plots shortcuts. The Micro Plot which is designed to print on an 8.5” x 11” piece of paper and the Big Plot which is designed to be plotted on a 36” x 48” sheet of paper. You can use these as is, modify them to suit your needs, or create new plots from scratch.

To create a new plot

1. Click the *Plots* shortcut tab.
2. Right-click in the *Plots* shortcut area and choose **New Plot**.
3. Enter the name of the new plot.
4. Click *OK*.

   Result: An empty plot is created. A shortcut to your new plot is added to the plots shortcut bar.

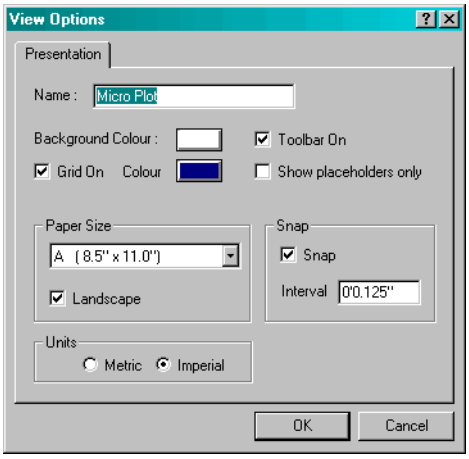
By default empty plots are setup to use 8.5” x 11” sheets of paper.

To set or change the paper size

1. Select the plot for which you want to set or modify the paper size.
2. From the **Options** menu, choose **View Options**.

   Tip: You can also click the *View Options* tool on the *Standard* toolbar.

   Result: The *View Options* dialog box is displayed.



3. In the *Paper Size* box, choose the desired paper size from the pull down menu. If you do not find the size you want, choose *Custom* and specify the width and height as desired.
4. Select the *Landscape* check box if you want the page to be setup in landscape format.
5. Click *OK*.

   Result: The gridded work area is resized accordingly.

Notes:

1. The gridlines are for reference only (and can be turned off); they will not print on your final output.
2. The solid outline shows your current printer's paper size and orientation settings. The large dashed line inside of that show the printable area for those settings.

Additional Working Preferences

Additional settings that appear on the *View Options* page are as follows. These additional settings will not affect the printout of the plot:

1. Click the box to the right of *Background Colour* to set the background colour of the plot view. This does not affect your printed copy.
2. Select the *Grid On* check box if you want gridlines to appear on the plot. Gridlines are for reference only; they will not appear in the printed output.
   - **a.** Click the box to the right of *Colour* to set the colour of the gridlines.
   - **b.** In the *Interval* box, specify the interval between the points on the grid.
   - **c.** Select the *Snap* check box to snap the items in the plot to the gridlines.
3. Select the units of measure for the plot. The coordinates at the bottom of the plot will change according to the selection.
4. Select the *Show placeholders only* check box to display only the placeholders for the various items in the plot. If not checked, both the placeholders and contents of the items in the plot are displayed. This is to facilitate the layout setup and does not affect the printout.

Inserting items onto plots

Objects can be inserted from the following categories:

- basic shapes (lines, rectangles, ellipses, polygons and text)
- bitmap images
- jpeg images
- plot items
- data items
- report items
- worksheet items (includes the default Title Block).

To insert basic shapes onto the plot

1 From the **Insert** menu, choose the appropriate command (**Line**, **Rectangle**, **Round Rectangle**, **Ellipse**, **Polygon** or **Text**).

Tip: You can also select the appropriate tool from the *Pres Insert* toolbar.

Line

Rectangle

Round Rectangle

Ellipse

Polygon

Text

2 Draw the item the same way you would in CAD mode. For more information on drawing, refer to "Drawing lines" on page 6.4-4.

To insert image, plot, data, report, or worksheet items onto the plot

1 From the **Insert** menu, choose the appropriate command (**Image**, **Plot**, **Data**, **Report** or **Worksheet**).

Tip: You can also select the appropriate tool from the *Pres Insert* toolbar.

Image Items

Plot Item

Spreadsheet Item

Report Item

Worksheet Item

2 Click and drag a window (placeholder) on the "page" where you want the item to appear.

3 If a dialog box appears (as in the case of image, data, report and worksheet items) choose the desired item and click *Select*.

Result: The object will be placed on the page in the placeholder drawn in step 2.

Content and Item properties

As with all objects in WYSIWYG, the objects you have placed on your plot have unique properties. There are two types of properties:

- Content Properties
- Item Properties

Content Properties

Content Properties affect the content of the presentation item. Opening these properties will open similar or identical dialog boxes as the *View Options* did when you were working on the original item in CAD or Data mode. In this dialog, you can modify the information to be displayed and control how it is displayed.

To modify content properties

1 Select the presentation item you want to modify.

2 From the **Edit** menu, choose **Content Properties**.

Tip: You can also click the *Content Properties* tool on the *Edit* toolbar.

Result: The *View Options* dialog is displayed.

Item Properties

Item Properties affect the placeholders and the scaling of the contents. Options here include the outline's weight and colour, and various printing options for the contents such as scale.

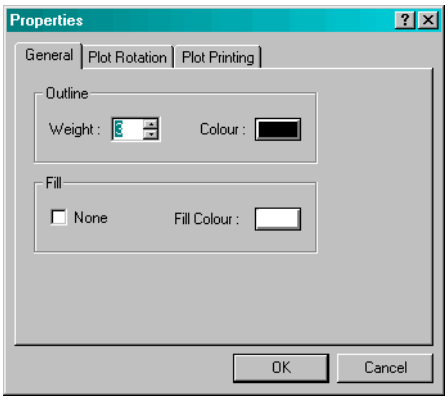
To modify item properties

1 Select the presentation item you want to modify.

2 From the **Edit** menu, choose **Item Properties**.

Tip: You can also click the *Item Properties* tool on the *Edit* toolbar.

Result: The *Properties* dialog is displayed.



Note: The first page of the properties dialog is identical regardless of the type of presentation item. Options here affect the outline and fill colour of the placeholder.

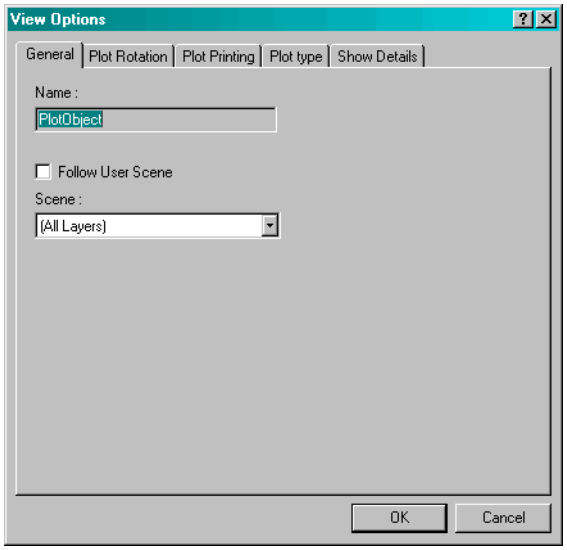
Plot items

Plot items are the most important objects on your plot. Their content properties are detailed below.

To modify content properties of plot items

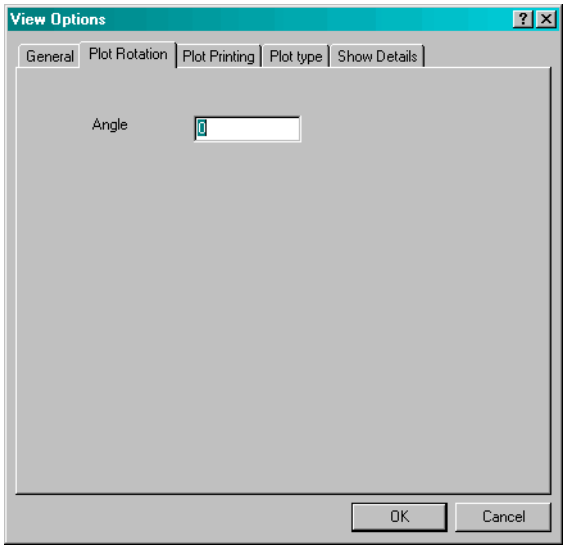
1 Select the plot item, right click, and choose **Content Properties** or click the appropriate tool on the *Pres Edit* toolbar.

Result: The *View Options* dialog box is displayed.



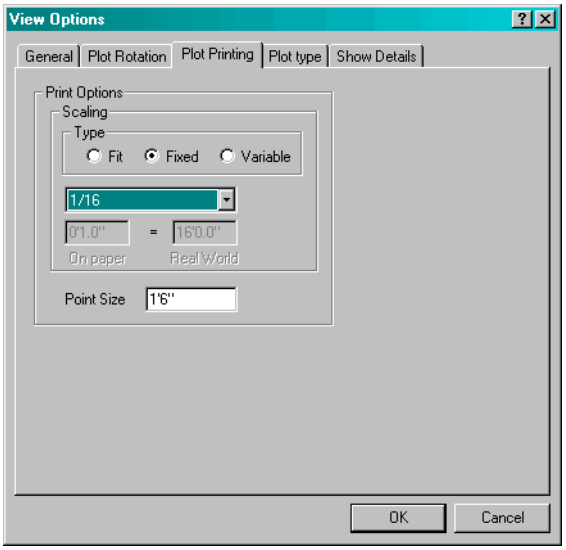
2 On the *General* tab, specify the scene to be used in the plot item. This will filter your show information based on the scene selected. Only fixtures and objects on the active layers will be displayed in the plot. For more information on scenes, refer to "Scenes" on page 6.5-15.

3 Click the *Plot Rotation* tab.



4 In the *Angle* box, enter an angle value in degrees if you want the drawing in the plot item to be printed on an angle. This feature is helpful for printing booms.

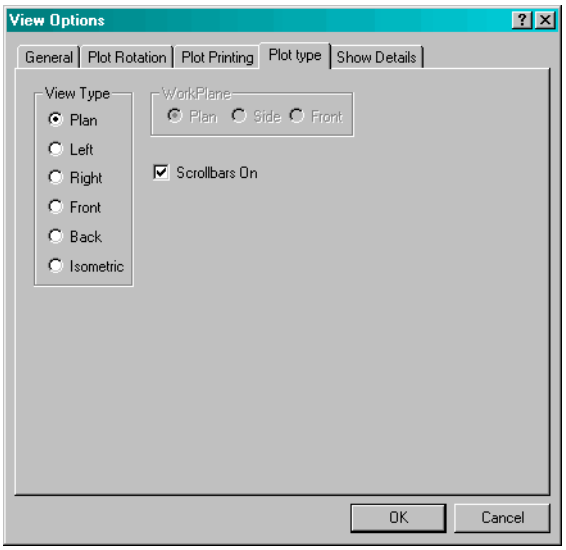
5 Click the *Plot Printing* tab.



6 Specify the scale for the drawing in this window. There are three types of scaling:

- Fit - The entire CAD drawing will be scaled to fit inside the placeholder.
- Fixed - The plot will be printed at the specified scale. To determine which portion of the plot is displayed inside the placeholder, chose **Open Contents**. Inside that view, you will be able to move around a white rectangle that represents the placeholder. Step 12 discusses the functionality available with Open Contents in further detail.
- Variable - In this case, when you "open contents" you will be able to change the size of the white rectangle without affecting the original placeholder. The portion of the drawing contained within the white rectangle will be scaled to fit in the placeholder. Step 12 discusses the functionality available with Open Contents in further detail.

7 Click the *Plot type* tab.

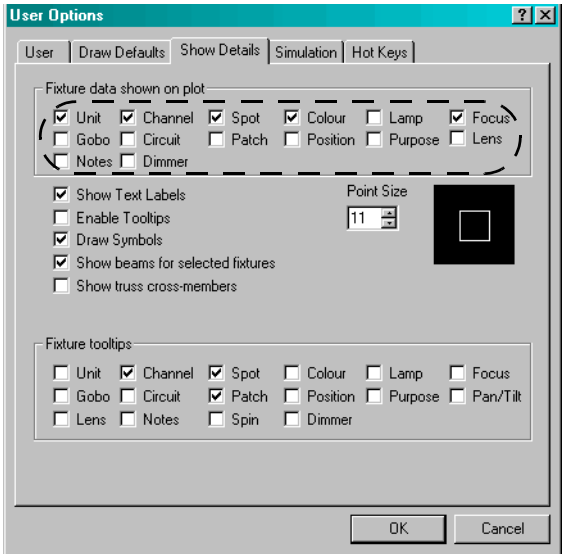


8 Choose the view type for the drawing in this window:

- Plan for a top view
- Right or Left for a section

- Front or Back for an elevation
- Isometric (Not available for WYSIWYG Report) for a perspective view. Note that isometric/perspective views cannot be scaled.

9 Click the *Show Details* tab.



10 In the *Fixture data shown on plot* box, select the attributes you want displayed around the fixture symbols on the plot. To modify how the attributes are laid out, refer to Fixture Notation and Information Layout on page 6.8-16.

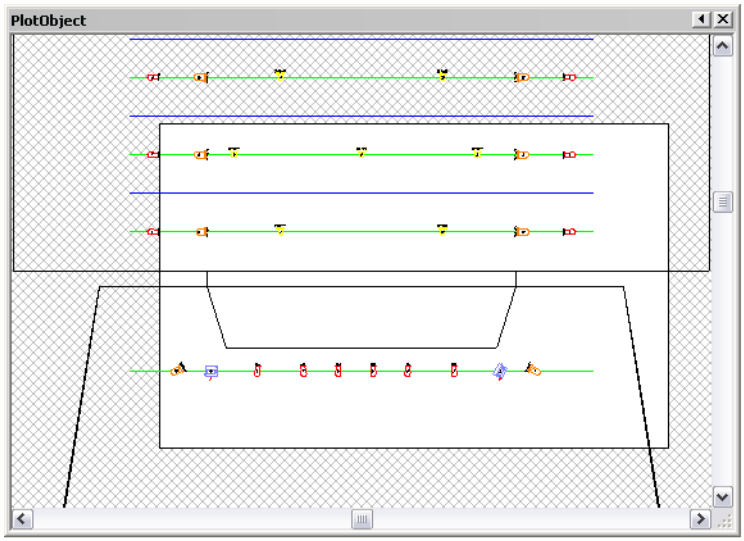
11 Select any other desired options and click *OK* to close the properties box.

12 If you chose fixed or variable scaling in step 6, you can further setup how the plot will appear within the placeholder by modifying the open contents:

a. With the plot item still selected, from the **Edit** menu, choose **Open Contents**.

Tip: You can also click the *Open Contents* tool on the *Edit* toolbar.

Result: A new window opens displaying your drawing and the plot item placeholder.



b. Adjust the positioning of the placeholder as desired.

c. Close the window.

Image Items

Special considerations should be given to printing images on plots. To maintain the integrity of the image, WYSIWYG can maintain the aspect ratio of the image so as not to distort it.

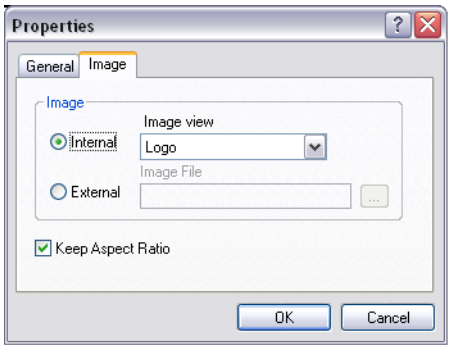
To maintain aspect ratio

1 Select the image item on the plot.

2 From the **Edit** menu, choose **Item Properties**.

Tip: You can also click the *Item Properties* tool on the *Edit* toolbar.

3 Click the **Image** tab.



4 Select the *Keep Aspect Ratio* check box.

Result: When checked, the height versus width relationship of the original image is maintained regardless of how you resize the placeholder.

Worksheet, Report, and Spreadsheet Items

Worksheet, Report, and Spreadsheet items are inserted into your plot at a set scale (100%) regardless of the paper size. For readability purposes, this value can be readjusted.

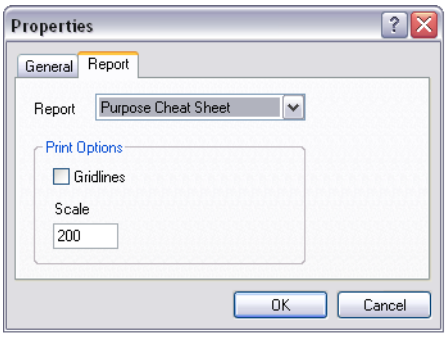
To adjust the scale

1 Select the worksheet, report, or spreadsheet item on the plot.

2 From the **Edit** menu, choose **Item Properties**.

Tip: You can also click the *Item Properties* tool on the *Edit* toolbar.

3 Click the *Worksheet*, *Report*, or *Spreadsheet* tab as appropriate.



4 Enter a percentage value in the *Scale* box to increase or decrease the size of the contents.

Plot Layout Tools

Placeholders

Placeholders can be moved and resized at any time.

To resize a place-holder

1 Click the presentation item to select it.

2 Click and drag the marker points of the placeholder to resize it.

Note: Resizing a placeholder will affect which portion of the content that is displayed. You may be required to edit the scale of the contents or revisit open contents.

To move a place-holder

1 Click the presentation item to select it.

2 When a small grey rectangle appears on your cursor, click and hold the left mouse button and drag the placeholder to the desired location.

3 Release the mouse button.

Tip: Use the gridlines of the presentation view to line up placeholders.

To copy a place-holder and its contents

1 Click the presentation item to select it.

2 When a small grey rectangle appears on your cursor, press and hold the CTRL key.

3 Click and hold the left mouse button and drag the copied item to the desired location.

4 Release the mouse button.

Ordering

To facilitate overlaying and underlaying placeholders, use the *Bring to Front* and *Send to Back* tools. Think of placeholders as being piled on top of each other. *Bring to Front* will move the selected placeholder and its contents to the top of the pile. *Send to Back* will move the selected placeholder and its contents behind all of the others.

To overlay a place-holder

1. Click the presentation item to select it.
2. From the **Edit** menu, choose **Bring to Front**.

   Tip: You can also use the *Bring to Front* tool on the *Pres Edit* toolbar.

To underlay a place-holder

1. Click the presentation item to select it.
2. From the **Edit** menu, choose **Send to Back**.

   Tip: You can also use the *Send to Back* tool on the *Pres Edit* toolbar.

9.4 Images

Images such as renderings and logos are stored in the Images section of the Presentation mode. Renderings are automatically saved here when "internal" is selected in the Output Properties/Step 2 of the Render Wizard. Any other images files (bitmap or jpeg format) may be imported.

To import a bitmap or a jpeg

1. Right-click on the *Images* shortcut tab and choose **New Image**.
2. Enter the name of your image.
3. Click *OK*.
4. Click on the shortcut for your new image. You should see "No Image Available" in the working area of the screen.
5. From the **File** menu, choose **Import**.
6. Use the browser to find the image file you want to import.
7. Click *Open*.

To export a bitmap or a jpeg

1. Select the image you want to export.
2. From the **File** menu, choose **Export**.
3. Use the browser to select a destination and a name for the image file you want to export. Ensure that either bitmap (.bmp) or jpeg (.jpg) is selected in the *Save As Type* box.
4. Click *Save*.

9.5 Worksheets

The Worksheet tab opens a spreadsheet not unlike many other spreadsheet programs. You can enter data and create mathematical functions here, as well as import and export data to other spreadsheet programs. This is useful for creating spreadsheets for crew requirements, schedules, budgets or any other show specific data.

To import data

1. Right-click on the *Worksheets* shortcut tab and choose **New Worksheet**.
2. Enter the name of your worksheet.
3. Click *OK*.
4. From the **File** menu, choose **Import**.
5. Use the browser to find the file containing the data you want to import.
6. Click *Open*.

To export data

1. Select the worksheet you want to export.
2. From the **File** menu, choose **Export**.
3. Use the browser to select a destination and a name for the worksheet file you want to export. Ensure that the desired export file type is selected in the *Save As Type* box.
4. Click *Save*.

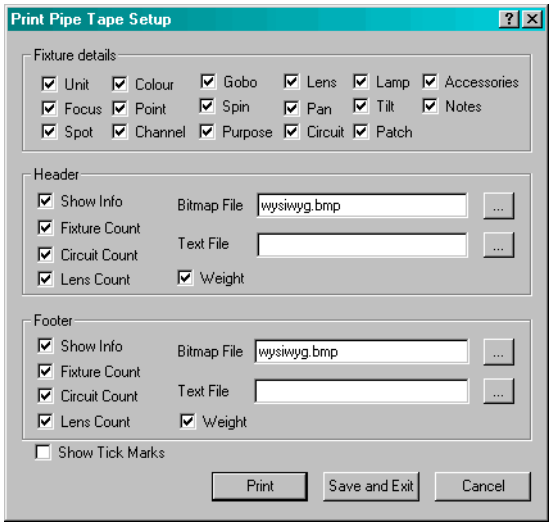
9.6 Pipe Tapes

WYSIWYG allows the user to make "Pipe Tapes". Pipe tapes are attached to a pipe at the beginning of a hang. The pipe tape indicates to the electricians where to hang the fixtures and displays pertinent information about the fixtures such as circuit #, dimmer #, gel colour, etc. as specified by the user. This tool significantly increases the accuracy between the WYSIWYG plot and the real installation. Pipe tape printers are sold separately and are available through your local ETC dealer.

To print a pipe tape

1. Connect the pipe tape printer to your computer via the serial port.
2. Return to the CAD mode and select the pipe for which you want to generate a pipe tape.
3. From the **Tools** menu, choose **Print Pipe Tape**.

   Result: The *Print Pipe Tape Setup* dialog box is displayed.

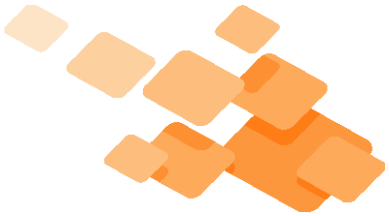


4. In the *Fixture details* group box, select the data fields that you want printed on the pipe tape for each fixture.
5. To print show information such as venue, lighting designer, and show name, select the *Show Info* check box. Show information is entered in Show options described on page 9.2-4.
6. To display a graphic in the header and/or footer, enter a file name in the *Bitmap File* box or use the file browser to locate the file. The graphics selected must be a monochrome bitmap in order to be displayed.
7. To print a total count of the number of fixtures on the selected pipe, select the *Fixture Count* check box. The total count is broken down by fixture type.
8. To print a total count of the number of circuits used by the fixtures, select the *Circuit Count* check box.
9. If you want to display text in the footer and/or header of the pipe tape, enter a text file name in the *Text File* box.

10. To print a total count of the number of lenses mounted in the fixtures, select the *Lens Count* check box. The total count is broken down by fixture type.
11. To print the weight of the fixtures on the pipe, select the *Weight* check box.
12. To display tick marks on the pipe tape, select the *Show Tick Marks* check box. Tick marks are placed every foot or half meter depending upon the unit of measurement (imperial or metric) for reference purposes.
13. When you have finished making your selections, click *Print* to send the print job to the pipe tape printer.
14. Click *Save and Exit* if you want to save the settings but do not want to print the pipe tape at this time.

Chapter 10.0

Live Mode



Live mode is used for graphically simulating the output of a lighting control console or compatible offline editor. Live mode also has rendering ability so that you can create photo-realistic pictures of the simulated lighting looks. Unlike Design mode, Live mode will display transitions from cue to cue, allowing you to see the programmed movement of light over time.

Live mode is available only in WYSIWYG Perform.

In this chapter

10.1 Layout Tabs	10.1-1
10.2 Connecting to Consoles	10.2-1
10.3 Autofocus	10.3-1
10.4 AutoPatch	10.4-1
10.5 Simulation Options	10.5-1
10.6 Rendering	10.6-1

10.1 Layout Tabs

Beneath the work area in each mode is a series of layout tabs. These layouts provide various configurations of the views you are working with. To change layouts, click the tab that corresponds to the layout you wish to use. The Live mode contains the following layout tabs:

- Wireframe - work area displays full screen wireframe view.
- Quad - work area is divided into quadrants, three of which can be modified to show plan, front, or side views. The lower-right quadrant contains a shaded view.
- Shaded - work area displays full screen shaded view.

Note: The wireframe views in Data mode are used for fixture selection only. To make any drawing modifications, you must return to CAD mode.

10.2 Connecting to Consoles

To use Live mode, you will need to insert and connect to a lighting control console or compatible offline editor. If you use a DMX console, a DMX interface device is required to connect to WYSIWYG.

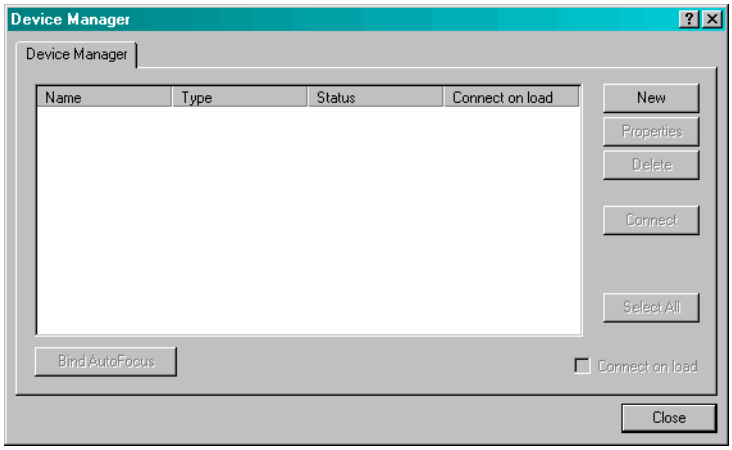
You will need to make sure that the patch you have created in WYSIWYG and the patch in your console are exactly the same for simulation to work properly.

Connecting to a console

To connect to a console

1. Connect the console to the computer with the appropriate hardware.
2. From the **Live** menu, choose **Device Manager**.

   Result: The *Device Manager* dialog box is displayed.



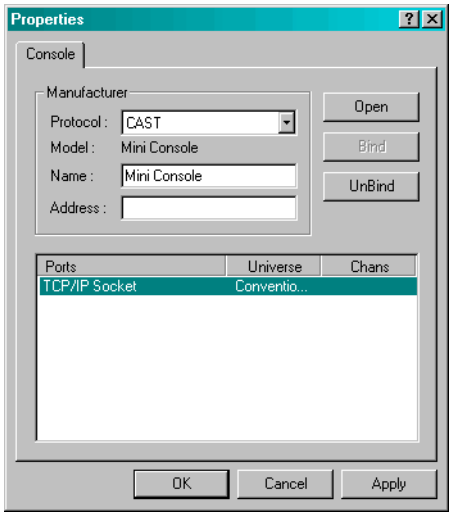
3. Click *New*.

   Result: The console library is displayed.
4. Navigate through the console library until you find the console or device you are connecting to. Click the console name to highlight it.

   Note: If you are connecting to a console using a Wyg-it or Wyg-it2, you can select either the console or the Wyg-it device. The *Address* box discussed below must contain the name of the Wyg-it.
5. Click *Insert*.

   Result: The console is displayed in the *Device Manager*.
6. With the console name still selected, click *Properties*.

   Result: The *Properties* dialog box is displayed.



7 Select the protocol used by the console from the *Protocol* drop-down list. For example, DMX, SandNet.

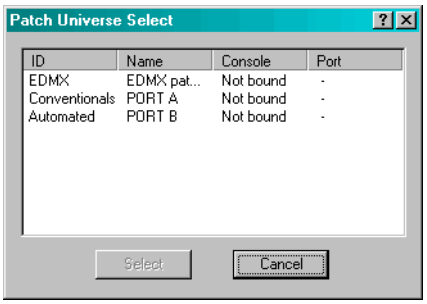
The console model is displayed in the *Model* box. The name of the console is displayed in the *Name* box.

8 In the *Address* box, enter the designated address of the console, if applicable.

9 In the *Ports* list, a list of the output ports from the console is displayed. You must bind these outputs to WYSIWYG patch universes. Click the appropriate port to highlight it.

10 Click *Bind*.

Result: The Patch Universe Select dialog box is displayed.



11 Select the desired Patch Universe from the dialog box that appears and click *Select*.

12 Repeat the steps above to bind all ports to their appropriate patch universes.

Note: Remember that you will not be able to control a channel count exceeding that of the WYSIWYG Perform package you purchased - 500, 1000, 2000, 5000, etc.

13 Click *OK* to close the *Properties* box.

14 Click *Connect*.

Result: The status of the console should change to "connected".

15 Click *OK* to exit the Device Manager.

Once the console is running and connected to the WYSIWYG drawing, operate the console as you would in the live venue. Run up channels and record cues using the console. Remember that WYSIWYG does not store or save cues. This is done in your console's memory.

Once the console information is setup, you can set WYSIWYG to connect to the console automatically everytime the file is opened.

To set the console to connect on load

1. From the **Live** menu, choose **Device Manager**.

   Result: The *Device Manager* dialog box is displayed.
2. Select the console.
3. Select the *Connect on load* check box.
4. Click *OK* to close the *Device Manager*.

To edit the patch binding settings

1. From the **Live** menu, choose **Device Manager**.

   Result: The *Device Manager* dialog box is displayed.
2. Select the console.
3. Click *Properties*.
4. Select the port for which you want to change the patch binding.
5. Click *Unbind*.
6. Rebind as desired (follow steps 9 through 12 in "To connect to a console" above).

To remove a console from the Device Manager

1. From the **Live** menu, choose **Device Manager**.

   Result: The *Device Manager* dialog box is displayed.
2. Select the console you want to remove.
3. Click *Delete*.

   Result: The console is removed; all patch binding information is lost.

Connecting with a Wyg-it 2

WYG-it 2 is Cast Software's DMX interface device. A DMX interface is required to connect a DMX console to WYSIWYG. Each WYG-it 2 has 2 DMX inputs allowing the user to visualize up to 1024 channels of DMX. The WYG-it 2 also has 2 DMX throughs enabling the user to connect the console to WYSIWYG and to the physical lighting equipment at the same time. WYG-it 2 is equipped with the necessary outputs to send Autofocus commands.

The WYG-it 2 connects to a user's computer via the USB port and does not need an external power supply. Connect the WYG-it 2 to your computer using a USB cable. For further instructions on setting up the Wyg-it 2, refer to the instructions available on the WYG-it 2 CD cover or the Readme file on the CD.

Note: Once the WYG-it2 is installed, the *WYG-it DMX Interface* configuration panel is accessible in the Control Panel of your computer. The WYG-it2 name recorded in this configuration panel is the value that should be entered in the *address* field of the *Properties* of the device in the *Device Manager*.

Mini Console

WYSIWYG contains an application that simulates a small console, called the Mini Console, that can be used in lieu of a real console. To use the Mini Console, run the Mini Console application, found in the WYSIWYG section of the Windows Start menu. Connect as if you are using an external console.

The Mini Console gives you control of the first 25 channels on a single DMX universe but does not record or playback cues.

10.3 Autofocus

Autofocus is the protocol that WYSIWYG uses to "talk back" to a compatible console. Using the mouse in a Wireframe view or using the concept shortcuts and design tools at any time, you can direct the console to select fixtures, set focus, and control intensity, iris, and colour. Autofocus is connected through MIDI, Serial, or Ethernet.

The following consoles are compatible with the Autofocus protocol:

- ETC Expression and Insight consoles
- Flying Pig WholeHog II
- Jands Hog 500, 1000, and Echelon consoles
- Compulite's Animator 48C, Animator 72C, and Sabre consoles
- Rosco Horizon software

To bind Autofocus to the console

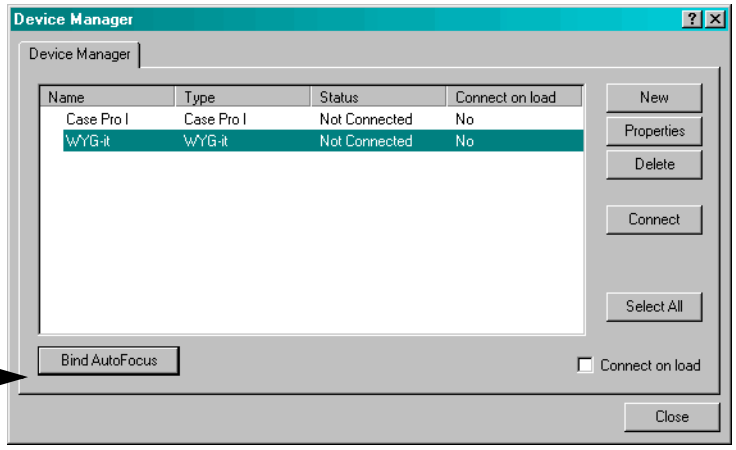
1 Follow steps 1 through 14 of "To connect to a console" on page 10.2-1.

2 Click *Bind Autofocus*.

Note: The *Bind Autofocus* button is only available when a compatible console is selected in the Device Manager.

3 The autofocus icon appears beside the name of the console in the Device Manager.

Bind Autofocus button



4 Click *Connect*.

5 Click *OK* to close the Device Manager.

Autofocus tools and menu commands

To select fixtures

The procedure for selecting fixtures in Live mode is the same as selecting fixtures in Design mode using the design tools. For more information, refer to "Using the Design tools" on page 8.2-2.

To use the Autofocus tools

The procedure for using the autofocus tools is the same as using the design tools. For more information, refer to “Using the Design tools” on page 8.2-2.

To use Autofocus menu commands

Right-click on a selected fixture or fixtures. This opens a menu with the following Autofocus commands:

- **At Level and Focus**: Sets the intensity of the selected lights on the connected Autofocus compatible console to the level last specified by the Intensity command and launches the Focus command.
- **Focus**: When you click in a Wireframe view, the selected automated fixtures will focus (if they can) to that point. Click and drag to position them in real time. Change the missing focus coordinate to alter the height at which you want to focus the fixtures.
- **Intensity**: Reveals a sub-menu where you can specify the level to set the intensity for the selected fixtures. The first option in the list is the last level you specified and by default is **Full**. You can also set them to half and out.
- **Iris**: Tells lights to either iris tight or wide. There is also a Specify option where you can set it to any percentage. The Specify value is remembered for the session.
- **Colour**: Fixtures that can mix colour can be directed to either a palette colour or an approximation of a manufacturer’s gel colour. Palette colours can be specified and saved by the user using the custom colour creator or Palette shortcuts.

Tips:

- Autofocus commands can be repeated by touching the Space bar. That way you can select one fixture, adjust it, select another, and then press the Space bar to perform the same operation repeatedly.
- Alter the missing focus coordinate to focus fixtures at different heights.
- When focusing fixtures, you can click and drag in the wireframe view. If you are using WYSIWYG in the theatre, launch this command, then look to the stage to position your lights.
- Working in an Isometric view you can use the snap tools to focus fixtures to points on your set pieces.
- If you set the intensity to a specific value, the right click menu option **At Full** and **Focus** will change to At ##% and Focus where ## is the specific value you set.
- Select multiple fixtures by dragging a window around them. If you drag the window from the right to left across projected beams, you can select the fixtures that produced the beams
- To select fixtures by type, draw a box using the right mouse button. You will be prompted to isolate one or more types of fixtures.
- Use the Concept shortcuts to create and select groups of fixtures.
- Use the Palette shortcuts to save and send colour commands.

10.4 AutoPatch

AutoPatch is a protocol by which WYSIWYG can transfer the patch hookup to a compatible console. AutoPatch is only available in WYSIWYG Perform.

You must be connected to a compatible console for this feature to be enabled. All ports for which you want the patch information to be transferred must be properly bound. Only the hookup information in patch universes bound to the console's outputs will transfer.

The patch information is transferred over Ethernet.

To perform an AutoPatch

1 Connect to the console as described in "To connect to a console" on page 10.2-1.

2 From the **Live** menu, choose **AutoPatch**.

Result: The *AutoPatch* dialog opens.

3 Click *Send Patch*.

Result: WYSIWYG assembles the patch information for all of the fixtures in the plot and then attempts to send the relevant patch information to each connected AutoPatch-capable device.

Note that a fixture's patch information will only be transmitted to a console if the fixture has at least one part that is patched to a patch universe that is bound to the console, and that console is connected (to WYSIWYG via the *Device Manager*).

It is possible for some fixtures to have parts patched to two (or more) different consoles. In this situation, both consoles will receive the patch information for that fixture, but the port numbers given for each part will be set appropriately for each console.

Note: It is recommended to click *Clear Patch* prior to sending the patch in order to clear out the existing patch.

4 Click *Close*.

To clear the patch

1 Connect to the console as described in "To connect to a console" on page 10.2-1.

2 From the **Live** menu, choose **AutoPatch**.

Result: The *AutoPatch* dialog opens.

3 Click *Clear Patch*.

Result: WYSIWYG will prompt each AutoPatch capable device to clear the patch for each port that is bound to a WYSIWYG patch universe.

4 Click *Close*.

10.5 Simulation Options

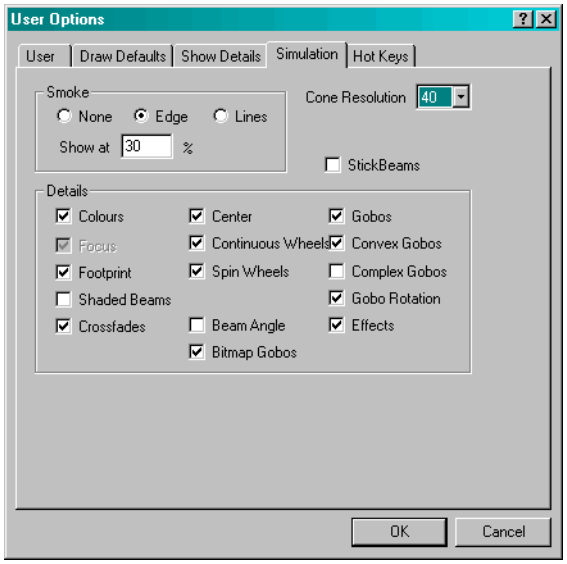
The Simulation options affect how fixtures respond in the Live mode and how the beams are displayed in shaded views. Turning options off will yield faster simulation performance while compromising aesthetics. For optimal results, increase the processing power of your computer.

To change the simulation options

1 From the **Options** menu, choose **User Options**.

Result: The *User Options* dialog box is displayed.

2 Click the *Simulation* tab.



3 Make your selections and click *OK* to exit.

a. The smoke level will affect the density of the cones of light.

b. The cone resolution affects the shape of the footprint. The higher the number the more elliptical, the lower the number the more square it will be. 5 is the fastest while 40 is a near perfect circle or ellipsis.

c. Select the *StickBeams* check box to draw the center of the light beam for each fixture. This is the quickest way to simulate moving lights with the least beam clutter.

d. Unselect the *Colours* check box if you want all beams to be displayed as open-white and ignore colour settings.

e. Select the *Focus* check box to set the image in focus as the brightest, in Wireframe views. In shaded views, the hard or soft edge can be seen when the focus parameter is set.

f. Select the *Footprint* check box to draw hard lines depicting where the light hits the set.

g. Select the *Shaded Beams* check box to produce soft edge beam footprints on the set, showing you the cosine distribution of your lights

in shaded views. The objects comprising the set must have their facets split. To enable this setting, select *Facet Splitting* from the *Object Settings* tab of Show Options. For more information on this tab, refer to "Object Settings" on page 5.2-8.

h. Select the *Crossfades* check box to show you the physical limitations of the motors of the fixtures or electronic accessories. This will be evident for devices such as moving heads or scrollers, which might not be able to move as fast as the console is changing values. Crossfades will also show you which fixtures use internal timing on a separate speed channel.

i. Select the *Center* check box to draw a point from the fixture to the point on the stage where the center of the beam hits. If the fixture has CMY colour mixing, the line will change colour to match the output from the console.

j. Select the *Continuous Wheels* check box to show you automated wheels rotating into place as they would on the real fixtures. The effect of the wheel rotating 360 degrees rather than snapping in steps will be shown.

k. Select the *Spin Wheels* check box to simulate spin wheel fixtures. Some automated fixtures have wheels that spin continuously. Gobo wheels often take a lot of processing power and can slow down the refresh rate of other fixture's updates unnecessarily.

l. Select the *Gobo* check box to simulate gobo details. Gobo wheels often take a lot of processing power and can slow down the refresh rate of other fixture's updates unnecessarily. If you are concerned about cue timing, set the scene first with gobos, and then turn off this option to obtain a faster refresh rate.

m. Select the *Convex gobos* check box to simulate convex gobo fixtures. Some complex gobos that include shapes similar to a "C" have to be pre-processed to make them appear correctly. This can often reduce the refresh rate.

n. Select the *Complex gobos* check box to simulate complex gobo fixtures. Most gobos are drawn in a complex and a simplified form. The complex forms are far too slow for real time simulation and should only be used in rendering.

o. To simulate gobo rotation, select the *Gobo Rotation* check box. This option can slow down the refresh rate substantially as gobos can be rotating even when the fixture's intensity is at zero.

p. To simulate effects such as prisms, select the *Effects* check box. This might increase the number of lines that WYSIWYG has to draw, thus slowing down processing power.

Also noteworthy are the properties of shaded views. These are discussed in "Shaded Views" on page 4.6-1.

10.6 Rendering

At any time in the Live mode, you can generate a rendering of a lighting cue or look. Note that renderings are of static looks. The Render Wizard will calculate the rendering based on the DMX values being received at the moment you click *Finish*.

To render a lighting look

1. Display the lighting look you wish to render in the working area of the screen.
2. From the **Design** menu, choose **Render**.

   Tip: You can also click the *Render Wizard* tool in the *Design* toolbar.

   Result: The Render Wizard will open and guide you through a series of options. Click *Next* and *Back* to navigate through the Render Wizard.
3. Click *Finish*.

   Result: WYSIWYG generates the rendering of your lighting look or cue. If the rendering is saved internally, it is available as an image in Presentation mode.

For more information on rendering, refer to “Rendering” on page 8.6-1.

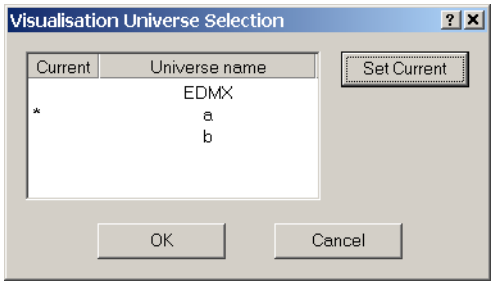
10.7 Visualization With WYSIWYG Perform - Console Edition

WYSIWYG Perform - Console Edition limits you to visualizing one patch universe at a time up to a maximum of 512 channels. Therefore, it is necessary for you to specify which universe to enable for visualization.

To select a patch universe for visualization

1 From the **Edit** menu, choose **Universes**.

Result: The *Visualisation Universe Selection* dialog box is displayed.



All the patch universes that are currently in the document are displayed.

2 Select the patch universe that you want to visualize and then click *Set Current*.

3 Click *OK* to close the dialog box.

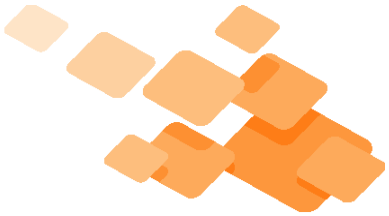
4 Repeat steps 1-3 to change the universe to be visualized.

Program your lighting console as normal considering that only fixtures patched in the selected universe will be simulated by WYSIWYG.

Tip:
There is a toolbar named *Universes* that gives you quick access to all existing patch universes. Select the universe you want to visualize from the drop-down list.

Chapter 11.0

WYSILink - Link Mode



The WYSILink (Link) mode becomes available when you purchase and install the WYSILink add-on for WYSIWYG. WYSILink acts as an interface to ETCLink. For further details on ETCLink, refer to your *Sensor CEM User Manual*.

In this mode you can view system diagnostic information about the fixtures in your system. You can also record and playback backup looks and view dimmer and dimmer rack information.

In this chapter

11.1 Layout Tabs	11.1-1
11.2 Setting up WYSILink	11.2-1
11.3 WYSILink Mode	11.3-1
11.4 ETCLink Functions	11.4-1

11.1 Layout Tabs

Beneath the work area in each mode is a series of layout tabs. These layouts provide various configurations of the views you are working with. To change layouts, click the tab that corresponds to the layout you wish to use. The Link mode contains the following layout tab:

- Message Log - work area displays a spreadsheet of ETCLink error messages.

11.2 Setting up WYSILink

WYSILink combines WYSIWYG visualization software with ETC's Sensor Advanced Features to perform system diagnostics. When Sensor detects a problem, WYSILink logs it, posts a detailed alert and highlights the image of the affected fixture along with lamp and dimmer data.

WYSILink highlights affected fixtures in Data and Wireframe views in CAD, Data, and Live modes, giving you the most complete information about the problem. You can also use the message log to track problems by fixture over time.

WYSILink resides on a PC connected to an ETCNet2 network and gives you access to all ETCLink features. You can record and play backup looks, view which console is controlling which dimmers, and obtain information from any dimmer or dimmer rack in your system.

Hardware requirements

WYSILink requires a few additional pieces of hardware in order to perform system diagnostics. For more information on hardware requirements, refer to "Installation" on page 3.1-1.

Upgrading to WYSILink

WYSILink is an optional feature set for WYSIWYG. You can upgrade your dongle for WYSILink operation by calling your ETC WYSIWYG dealer. Once you have purchased the upgrade, you will be provided with an upgrade code. To learn how to upgrade to WYSILink, refer to "Upgrades" on page 3.3-1.

ETCNet2 compatibility

WYSILink now supports ETCNet2 version 2.1.1 and version 3.0.0 networks. WYSILink can only monitor one ETCNet2 version at a time. By default, WYSILink will monitor an ETCNet2 version 2.0.0 network.

To set the version that is monitored, refer to "ETC Interface" on page 5.2-6.

11.3 WYSILink Mode

WYSILink mode is accessed through the *LINK* mode button at the top of the WYSIWYG screen. You can view the WYSILink Message Log and configure your WYSILink settings from the LINK mode.

Note: If you cannot see the *LINK* mode button you are not running WYSIWYG with a WYSILink-enabled dongle. Contact your dealer or ETC Technical Support for further information.

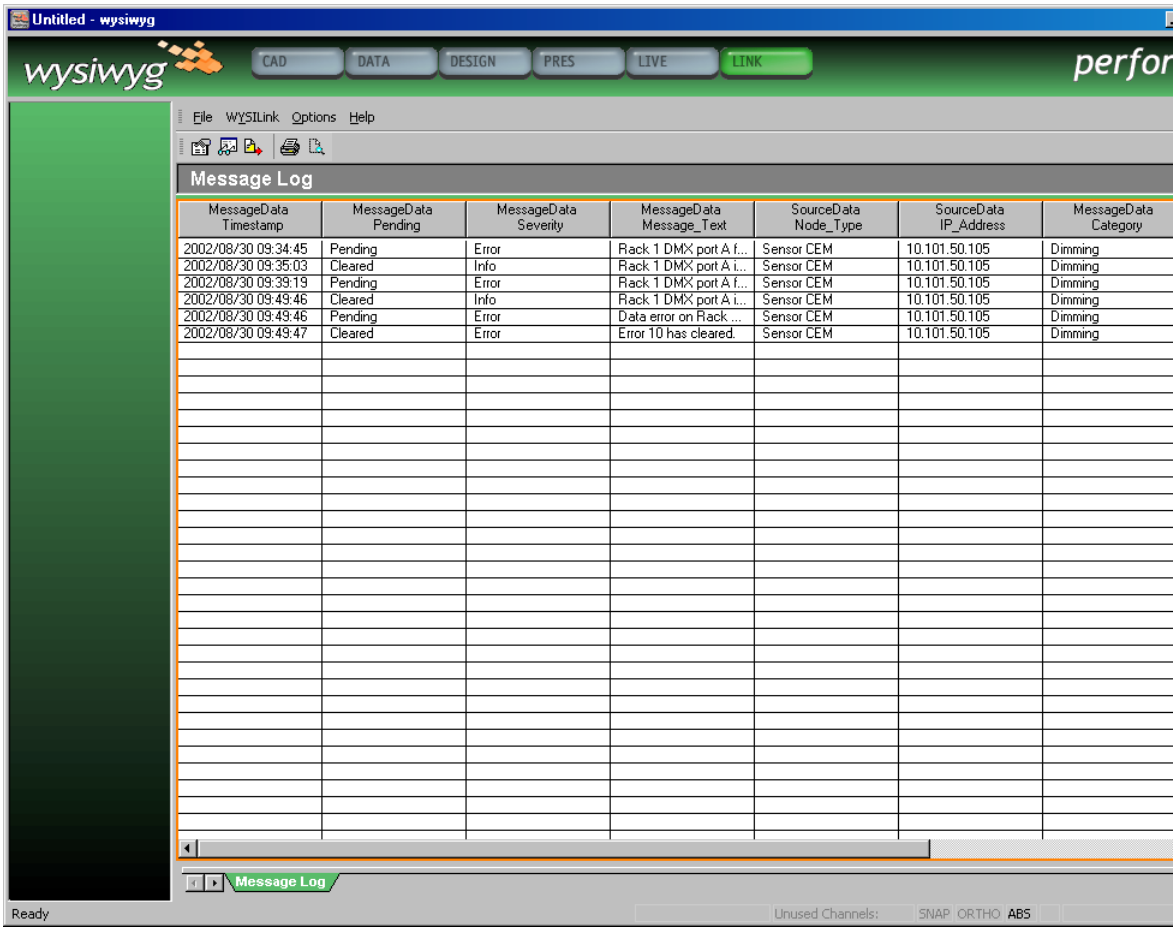
Message Log

The Message Log provides a spreadsheet view of ETCLink error messages. As error states occur, messages are logged in the Message Log, and classified as either "Pending" or "Cleared".

- Pending messages are generated at the time of the error.
- Cleared messages are generated when an error has been corrected.

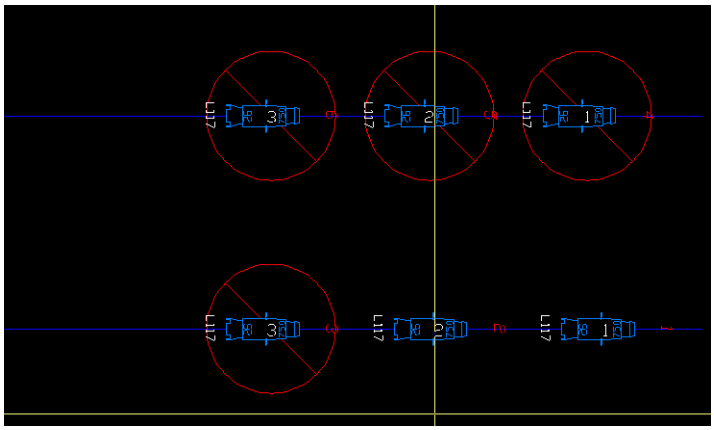
Pending and Cleared messages are paired, so that when an error state has been corrected, the appropriate Pending message is cancelled. You have the option of viewing all Pending and Cleared messages, or only Pending messages (for error states that have not been corrected).

Note: WYSILink defaults to displaying only Pending Messages. To change this setting refer to "Message Log options" on page 11.3-2.


Untitled - wysiwyg
wysiwyg
CAD
DATA
DESIGN
PRES
LIVE
LINK
perfor
File WYSILink Options Help
Message Log
MessageData Timestamp
MessageData Pending
MessageData Severity
MessageData Message_Text
SourceData Node_Type
SourceData IP_Address
MessageData Category
2002/08/30 09:34:45 Pending Error Rack 1 DMX port A f... Sensor CEM 10.101.50.105 Dimming
2002/08/30 09:35:03 Cleared Info Rack 1 DMX port A i... Sensor CEM 10.101.50.105 Dimming
2002/08/30 09:39:19 Pending Error Rack 1 DMX port A f... Sensor CEM 10.101.50.105 Dimming
2002/08/30 09:49:46 Cleared Info Rack 1 DMX port A i... Sensor CEM 10.101.50.105 Dimming
2002/08/30 09:49:46 Pending Error Data error on Rack ... Sensor CEM 10.101.50.105 Dimming
2002/08/30 09:49:47 Cleared Error Error 10 has cleared. Sensor CEM 10.101.50.105 Dimming
Message Log
Ready
Unused Channels:
SNAP ORTHO ABS

Special circumstances arise when errors are generated when WYSILink is offline. If an error is generated and corrected while WYSILink is offline, those messages will not appear in the Message Log. If an error has been generated when WYSILink is offline, and the message is not cleared, the Pending message will appear in the Message Log when WYSILink comes online.

Errors are indicated not only in the Message Log, but also in the Wireframe and Data views. Fixtures assigned to dimmers that have an error are highlighted in red in the spreadsheet, and by a red symbol in Wireframe views.



Message Log options

Message log options affect how messages are processed and received.

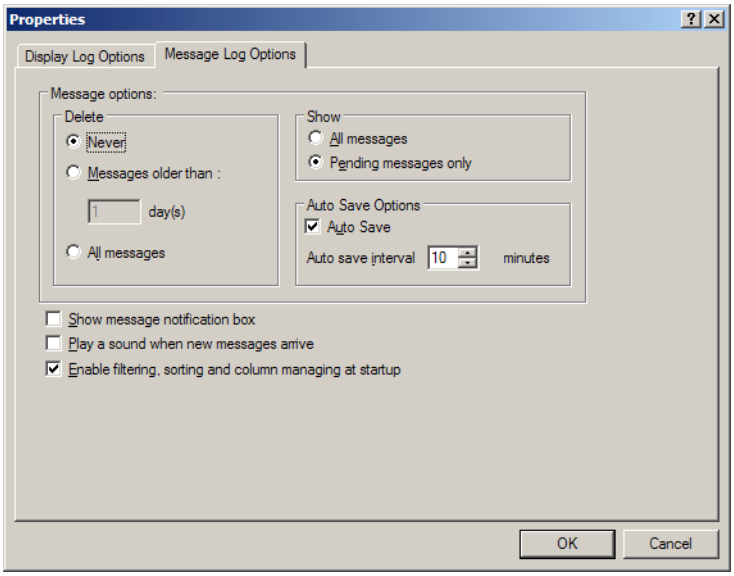
To access message log options

1 From the **WYSILink** menu, choose **Message Log Options**.

Tip: You can also use the *Message Log Options* tool on the *Message Log Options* toolbar.

Result: The *Properties* dialog box is displayed.

2 Click the *Message Log Options* tab.



Use the *Message Log Options* to configure the Message Log:

- Use the *Delete* settings to determine how long messages are held in the Message Log and to delete messages currently in the Message Log.
- Use the *Show* settings to switch between display of all messages (Pending and Cleared) or only Pending messages.
- Use the *Auto Save* settings to determine how often the Message Log is backed up to disk.
- Use the notification settings to setup how you want to be notified of new message arrival. You can choose to have a notification box pop-up, and/or have a sound play, as new messages arrive.

Display Log Options

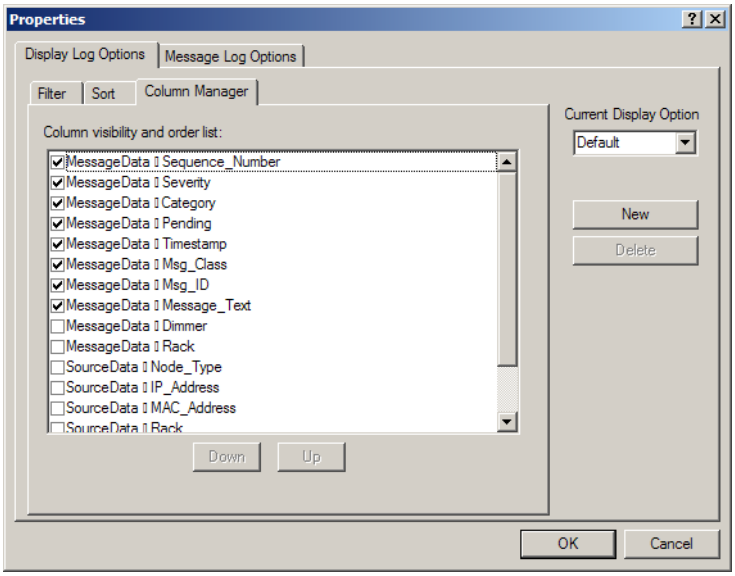
Display log options affect how columns are ordered and how the messages are sorted.

To set column visibility and order

1 Click the *Display Log Options* tool on the *Message Log Options* toolbar.

2 Click the *Display Log Options* tab.

3 Click the *Column Manager* tab.



4 Click the name of the column you want to move and click *Down* or *Up* to reposition it in the listing.

5 Click the box to the left of the column name to set its visibility. A check indicates the column is currently visible.

Note: The column labeled *Message Data Sequence Number* may not be made invisible.

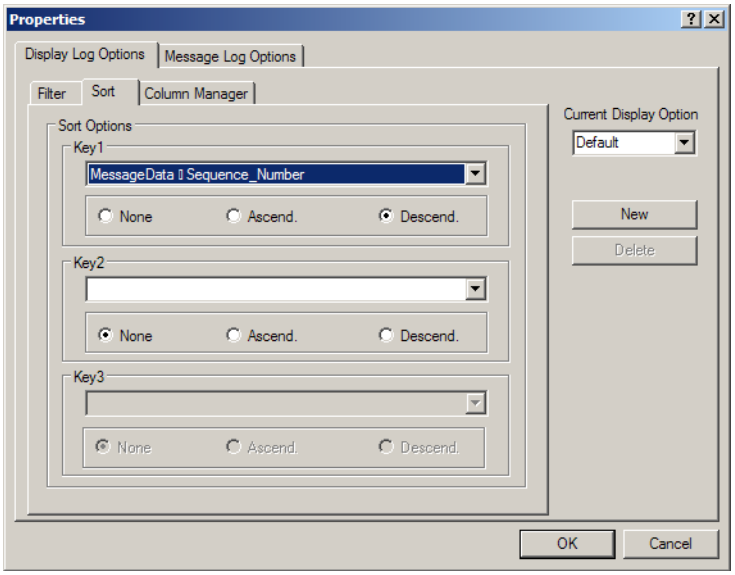
6 Click *OK*.

To sort messages

1 Click the *Display Log Options* tool on the *Message Log Options* toolbar.

2 Click the *Display Log Options* tab.

3 Click the *Sort* tab.



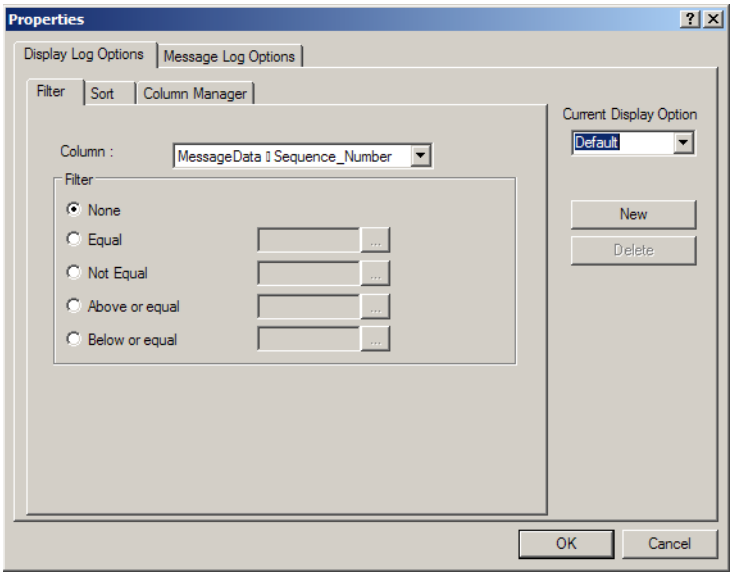
4 In each *Key* position, select the column you want to sort by, and click *Ascending* or *Descending* to set the sort order.

5 Click *OK*.

Result: The Message Log will refresh with the newly sorted data.

To filter messages

1. Click the *Display Log Options* tool on the *Message Log Options* toolbar.
2. Click the *Display Log Options* tab.
3. Click the *Filter* tab.



4. Select the column you want to filter.
5. Set the type of filter using the options.
6. Click *OK*.

   Result: The Message Log will refresh with the newly filtered data.

Exporting the Message Log

You can export the message log to a comma-delimited text file. This file can be viewed in most spreadsheet applications. This feature is useful if you are having system problems and need to get the Message Log to ETC Technical Services.

To export the Message Log

1. Click the *Export Message Log* tool on the *Message Log Options* toolbar.
2. Use the browser to set the name and destination of the exported file and click *Save*. The file should have the ".log" extension.

Printing the Message Log

To print the message log

1. From the **File** menu, choose **Print**.

   Tip: You can also use the *Print* tool on the *Message Log Options* toolbar.

To print preview the message log

1 From the **File** menu, choose **Print Preview**.

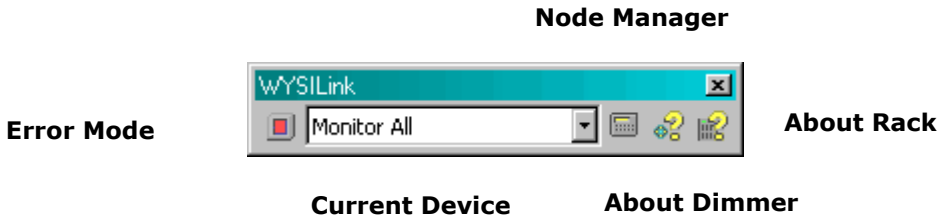
Tip: You can also use the *Print Preview* tool on the *Message Log Options* toolbar.

11.4 ETCLink Functions

WYSILink allows you to use many of the ETCLink features commonly found on ETC consoles. These features allow you to get information about your Sensor dimming system. You can access information about individual dimmers and about entire dimmer racks. You can also record dimmer loads and record and playback backup looks from within WYSILink.

The WYSILink toolbar

The WYSILink toolbar provides quick access to the ETCLink functions described below.



- Error Mode toggles between the currently selected node and all connected nodes.
- Node Selection allows you to view only the channels owned by that node.
- Node Manager (refer to "Node Manager" on page 11.4-3).
- About Dimmer (see "About dimmer" on page 11.4-1).
- About Dimmer Rack (see "About dimmer rack" on page 11.4-2).

About dimmer

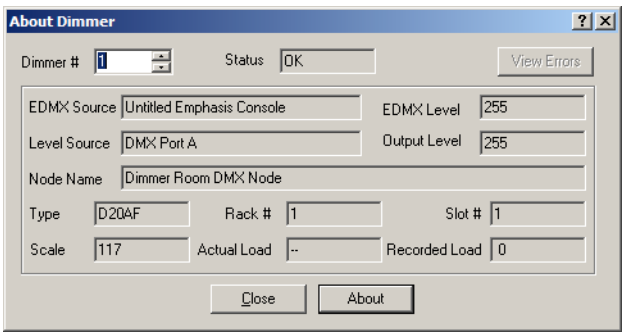
The *About Dimmer* dialog box provides specific information about an individual dimmer, such as the rack number, slot number, and type of dimmer. The name of the connected node, EDMX source, and the source for setting the level are also displayed.

To access dimmer information

1 From the **WYSILink** menu, choose **About Dimmer.**

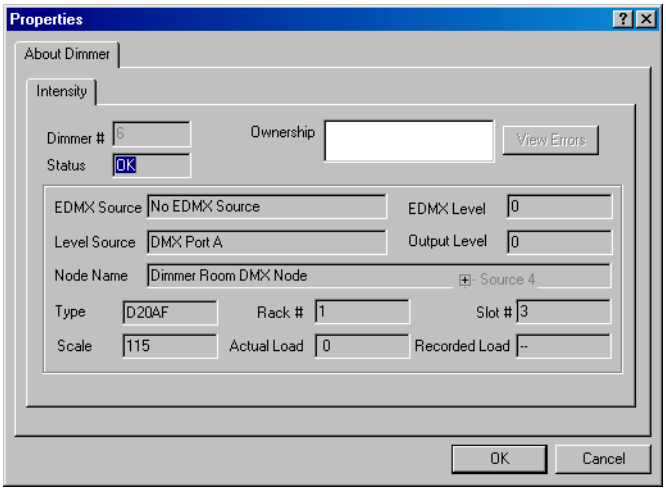
Tip: You can also use the *About Dimmer* tool on the WYSILink toolbar.

Result: The *About Dimmer* dialog box is displayed.



2 To obtain dimmer information, enter or scroll to the dimmer number, and then click *About*.

Tip:
You can also access the **About Dimmer** command in CAD, Data, and Live modes by right-clicking on a fixture symbol. This will open a slightly different dialog box.



Note: The *Ownership* box lists all network devices (such as consoles and nodes) that are attempting to control the specified dimmer.

About dimmer rack

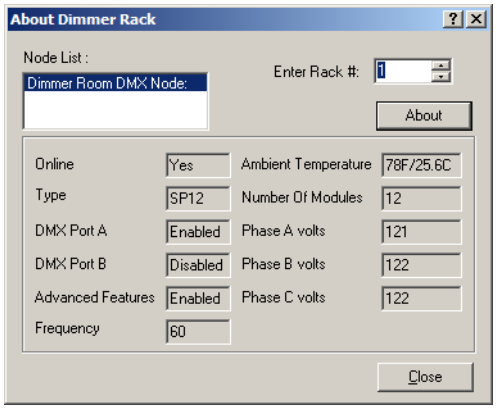
The *About Dimmer Rac*k dialog box allows you to choose a specific dimmer rack on a specific ETCNet2 DMX node. The status of the input ports (A and B), number of dimmer modules on the rack, and the dimmer type are displayed on this dialog box.

To access rack information

1 From the **WYSILink** menu, choose **About Rack**.

Tip: You can also use the *About Rack* tool on the WYSILink toolbar.

Result: The *About Dimmer Rack* dialog box is displayed.



2 Select a node, the rack number, and then click *About.*

Node Manager

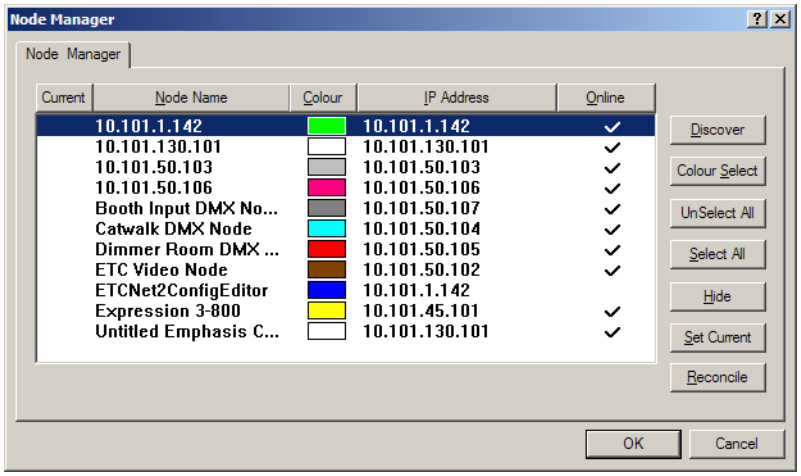
WYSILink adds the Node Manager tab to the Device Manager dialog box in Live mode. In this dialog box, you will need to "discover" connected nodes before other functions will work.

To access the Node Manager

1 From the **WYSILink** menu, choose **Node Manager**.

Tip: You can also use the *Node Manager* tool on the WYSILink toolbar.

Result: The *Node Manager* dialog box is displayed.



2 Click the *Discover* button to view the nodes that are currently online and set a colour for the display of channels "owned" by each node.

3 To set the colour for a node, double-click the colour field to the right of the node's name, or click the node's name and click *Colour Select*.

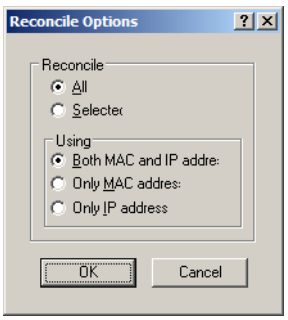
4 To remove an entry from the list, click *Hide.*

You can use the Node Manager to reconcile the configurations between your show document and the actual ETCNet2 Nodes connected to your ETCNet2 network.

To reconcile nodes

1 In the *Node Manager* dialog box, click *Reconcile*.

Result: The *Reconcile Options* dialog box is displayed.



2 Select whether you want to reconcile all nodes, or only currently selected nodes. Use CTRL+Click to select multiple nodes in the Node Manager.

3 Select whether you will use Media Access Control (MAC) addresses or Internet Protocol (IP) addresses.

4 Click *OK*.

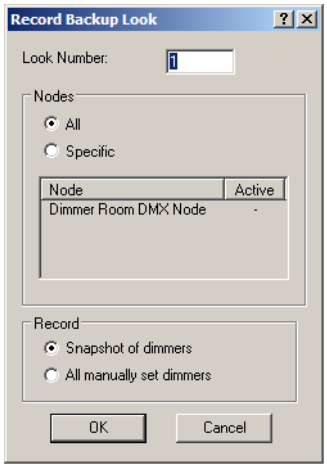
Backup looks

You can record, playback, stop and clear backup looks from within WYSILink. For more information on backup looks, refer to your *Sensor CEM User Manual*.

To record a backup look

1 From the **WYSILink** menu, choose **Backup Looks**, and then choose **Record**.

Result: The *Record Backup Look* dialog box is displayed.



2 Enter a number for the backup look that you want to record in the *Look Number* box.

3 Select the node(s) for which you want to record the backup look. Click *Specify* to select the node from the *Node* list. *All* indicates that the backup look will be recorded for all connected nodes.

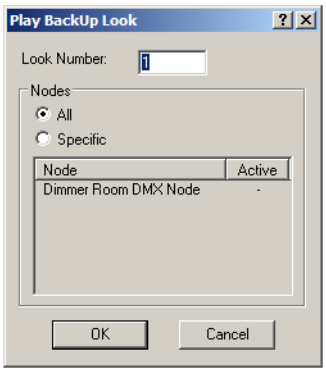
4 When recording a backup look, you have the option of recording the output level of all dimmers in the selected dimmer rack (*Snapshot of dimmers* option) or only those that have been set using the dimmer rack control panel (*All manually set dimmers* option).

5 When finished, click *OK*.

To playback a backup look

1 From the **WYSILink** menu, choose **Backup Looks**, and then choose **Play**.

Result: The *Play BackUp Look* dialog box is displayed.



2 Enter the number of the backup look that you want to play in the *Look Number* box.

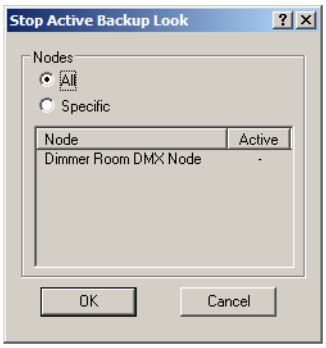
3 Select the node(s) for which you want to play the backup look. Click *Specify* to select the node from the *Node* list. *All* indicates that the backup look will be played for all connected nodes.

4 When finished, click *OK*.

To stop a playback look

1 From the **WYSILink** menu, choose **Backup Looks**, and then choose **Stop**.

Result: The *Stop Active Backup Look* dialog box is displayed.



2 Enter the number of the backup look that you want to stop in the *Look Number* box.

3 Select the node(s) for which you want to stop the backup look. Click *Specify* to select the node from the *Node* list. *All* indicates that the backup look will be stopped for all connected nodes.

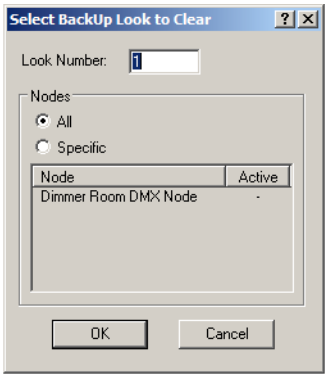
4 When finished, click *OK*.

To clear a backup look

Perform this procedure if you want to permanently erase a backup look from a dimmer rack.

1 From the **WYSILink** menu, choose **Backup Looks**, and then choose **Clear**.

Result: The *Select BackUp Look to Clear* dialog box is displayed.



2 Enter the number of the backup look that you want to clear in the *Look Number* box.

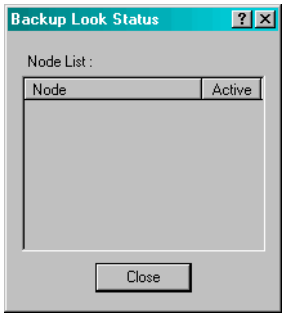
3 Select the node(s) for which you want to clear the backup look. Click *Specify* to select the node from the *Node* list. *All* indicates that the backup look will be cleared for all connected nodes.

4 When finished, click *OK*.

To obtain a backup look status

1 From the **WYSILink** menu, choose **Backup Looks**, and then choose **Status**.

Result: The *BackUp Looks Status* dialog box is displayed.



2 View which ETCLink-enabled DMX nodes have backup looks associated with them and if the looks are currently active.

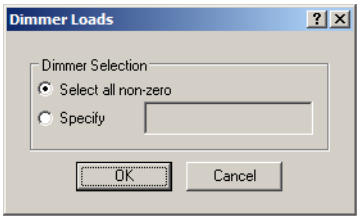
Note: Changes to backup looks will not be immediately visible. You may need to wait up to 30 seconds for the change to take effect.

Load recording

To record a load

1 From the **WYSILink** menu, choose **Load Recording**, and then choose **Record**.

Result: The *Dimmer Loads* dialog box is displayed.



2 Specify the specific dimmer for which you want to record a load or specify all non-zero level dimmers.

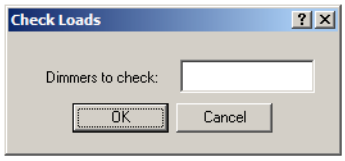
Recording the load on a dimmer will enable WYSILink to notify you when that dimmer's load has changed, such as when a lamp fails.

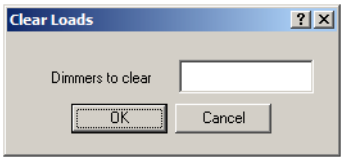
3 When finished, click *OK*.

To check or clear a load

1 From the **WYSILink** menu, choose **Load Recording**, and then choose **Check** or **Clear**.

Result: The *Check Loads* or *Clear Loads* dialog box is displayed.





2 Enter the number of the dimmer(s) to check or clear. Ranges may be specified using a dash, for example, 23-47. Use commas to separate the entries. For all dimmers specify "all". White space and capitalization is ignored.

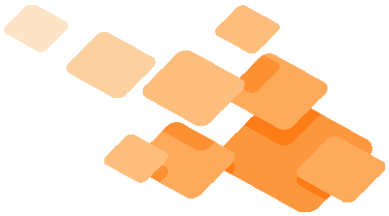
If you are checking a load, a query is performed for the specific dimmer or dimmers and a comparision between the recorded load and the actual load is conducted.

If you are clearing a load, the recorded load information is cleared for the selected dimmers.

For additional information on working with recorded loads, please refer to your *Sensor CEM User Manual*.

Appendix A.0

Using the SuperPro Monitoring Tool



The Sentinel SuperPro Monitoring Tool is installed as part of the WYSIWYG Learn and WYSIWYG Network Server installation. Using this tool, you can view and track license distribution information for the dongle(s) located on your network.

In this chapter

A.1 SuperPro Monitoring Tool A.1-1

A.1 SuperPro Monitoring Tool

Introduction

As part of the WYSIWYG Learn or WYSIWYG Network Server installation, a Sentinel SuperPro Monitoring Tool is included that allows you to view and track information about the dongle(s) on your network. Using this tool you can:

- keep track of how many licenses are in use on your network.
- view the maximum number of licenses that can be obtained from each dongle and therefore the maximum number of users (both local and on the network) that can access WYSIWYG Learn or WYSIWYG Network.
- view the SuperPro servers currently installed and running on your network.
- view the addresses and version information for each SuperPro server.

The Monitoring tool is a convenient tool to use to keep track of license activity on your network. For example, you can use the Monitoring Tool to determine whether you have purchased enough licenses, based on license demand.

Note: The SuperPro Monitoring Tool is designed for use only with SuperPro-protected applications, such as WYSIWYG Learn or WYSIWYG Network, that are being run on a network.

To launch the Monitoring Tool

1. Navigate to the folder where you installed the Server (for example, C:\Program Files\WYSIWYG Learn).
2. Open the Bin folder.
3. Double-click the *monitor* application.

   Result: The Monitoring Tool is launched, and you are prompted to set a protocol. The Monitoring Tool can monitor servers on the network that use any of the following protocols:

   - TCP/IP
   - NETBEUI
   - IPX/SPX
   - SAP
4. Select TCP/IP and then click *OK*.

   Result: Servers using the selected protocol are displayed in the Monitoring Tool. WYSIWYG products run on TCP/IP networks only.

Changing the protocol

When you launch the SuperPro Monitoring Tool, you are prompted to set a protocol.

To modify the selected protocol

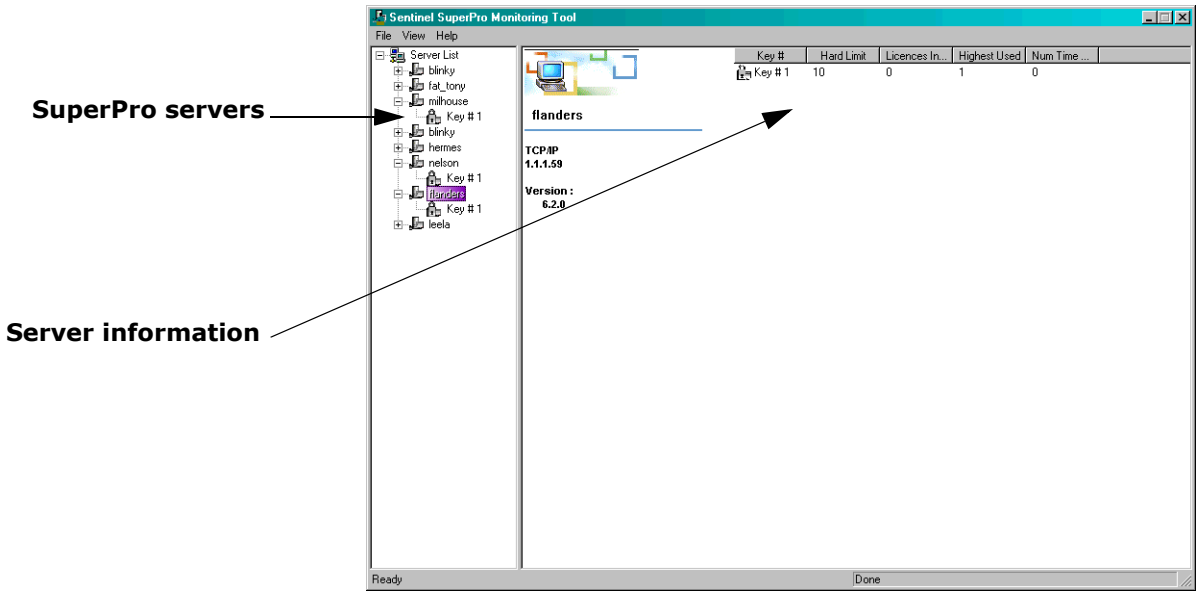
1. From the **File** menu, choose **Set Protocol**.
2. Select the protocol you want to use.
3. Click *OK*.

   Result: Servers using the selected protocol are displayed in the Monitoring Tool.

Viewing servers on the network

When you launch the Monitoring Tool, all SuperPro servers running on the network appear in the server list on the left. Clicking on a server in the list displays the following information on the right pane:

- server name
- server's protocol
- server's address (for TCP/IP or IPX/SPX protocols)
- server's version
- key information for the dongle that is connected to the server



Key information

Information about each key (or dongle) connected to a server running on the network appears in the right pane when you select a server from the server list. Click on a key to obtain the following information:

- key number (a label used only for identification purposes)
- key's hard limit (that is, the maximum number of licenses that can be obtained from the dongle and therefore the maximum number of users that can access WYSIWYG Learn or WYSIWYG Network.)
- number of licenses currently in use on the dongle
- highest number of licenses issued from this dongle
- number of time-outs recorded by the Server for Clients using licenses from the dongle.

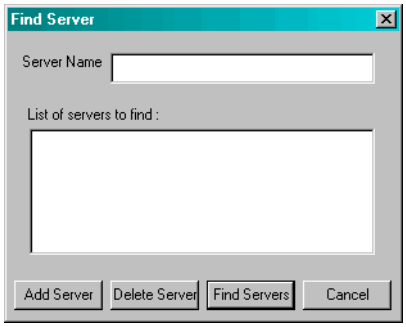
Searching for a server

If a server does not appear in the list, you can search for it on the network.

To search for a server

1 From the **View** menu, choose **Find**.

Result: The *Find Server* dialog box is displayed.



2 In the *Server Name* box, enter the host name or IP address of the server you are looking for.

3 Click *Add Server* to add the server to the list of servers to search for.

4 Click *Find Server* to search for the server(s). If the server is found, it appears in the server list in the left pane.

5 Once a server has been added to the list of servers to search for, it will remain on the list until you remove it. To remove a server from the list, select the server and then click *Delete Server*.

Refreshing the server list

You can either manually or automatically refresh the server list using the Monitoring Tool.

To manually refresh the server list

1 From the **View** menu, choose **Refresh**.

To auto-matically refresh the server list

1 From the **View** menu, choose **Refresh Rate.**

2 In the *Refresh After* box, enter the refresh rate in minutes.

3 Click *OK*.

Additional help

For further assistance in using the Sentinel SuperPro Monitoring Tool, refer to the online help available with the product.

Index

Numerics

2-way ortho, 5.2-13
4-way ortho, 5.2-13

A

Absolute coordinates, 4.4-1, 6.5-2
Acceleration, 5.2-3
Accessory, 6.9-1
Activation, 3.2-1
Ambient light, 4.6-3
Antialiasing options, 8.6-9
Aperature, 8.6-3
Application Options, 5.2-1
Arc, 6.4-6
Arena, 6.4-3
Arranging shortcuts, 4.2-3
Array, 6.6-8
Aspect ratio, 8.6-3, 9.3-8
Assembly Snap, 6.2-7, 6.7-7
Atmospheric options, 8.6-4
Attributes (Data fields), 7.2-1
Autofocus, 10.3-1
Auto-increment, 6.9-6
AutoPatch, 10.4-1
AutoUnit, 5.2-9, 6.7-1, 6.7-2

B

Back view, 6.2-5
Backup looks, 11.4-4
Barndoors, 6.9-1
Baud rate, 5.2-2
Beam simulation quality, 4.6-3
Beams, 5.2-14
Big Plot, 9.3-1
Black Box, 6.4-3
Booms, 6.7-6
Borders and legs, 6.4-9
Break tool, 6.6-2

C

CAD environment, 6.3-1
CAD Options toolbar, 4.5-2
Cameras, 4.6-3, 6.4-9
Center line, 6.4-4
Circle, 6.4-5
Circuit Name data field, 7.2-1
Circuit Number data field, 7.2-1
Circuit Type data field, 7.2-2
Close the show document, 5.1-4
Colour, 6.9-2
Colour data field, 7.2-1
Colour wheel, 8.2-3
Colour/gobo lists, 6.9-2
Columns in Data mode, 7.2-1
ComEdit toolbar, 4.5-2
Compass, 6.4-10
Concept shortcuts, 8.3-1
Connecting
  console, 10.2-1
Console data field, 7.2-2
Consoles
  connecting, 10.2-1
Contact ETC, 1.2-1
Content properties, 9.3-4
Convert tool, 6.6-2
Co-ordinate display, 5.2-2
Coordinate notation, 6.2-1
Coordinates, 4.4-2, 6.2-1
Cost data field, 7.2-2
Count column, 9.2-2
Creating a new DMX universe, 7.3-1
Creating a new spreadsheet, 7.2-7
Custom, 6.9-4
Custom fixtures, 6.8-2
Custom library items, 6.3-3, 6.4-10
Custom settings, 5.2-1
Customizing spreadsheet views, 7.2-2
Cylinders, 6.4-8

D

Data error tab, 5.2-10
Data fields, 7.2-1
Data filters, 7.2-5
Data toolbar, 4.5-3
Decimal display, 5.2-2
Design toolbar, 4.5-3
Design tools, 8.2-1
Designer's Remote, 10.3-2
Detail level, 9.2-2
Device Manager, 10.2-1
Dimension line, 6.4-7
Dimmer data field, 7.2-1
Dimmer information, 11.4-1
Dimmer rack information, 11.4-2
Direct 3D tab, 5.2-3
Display Log, 11.3-3
Distance tool, 6.6-9
Divide tool, 6.6-4
DMX, 10.2-1, 10.2-3, 10.2-4
DMX patch, 6.5-5
Document summary, 5.2-7
Dot line, 6.4-4
Draw defaults, 5.2-13
Draw toolbar, 4.5-4
Drawing, 6.4-1, 6.4-4, 6.7-3

Drawing pipes, 6.7-3
Drawing tips, 6.4-1
DWG, 5.1-1
DXF, 5.1-1
DXF/DWG files, 6.2-8

E

Edit (Pres) toolbar, 4.5-5
Edit toolbar, 4.5-4
Editing objects, 6.5-1
EDMX, 5.2-6
EDMX address, 6.8-14
Elements, 6.5-10
Entering data in the spreadsheet, 7.2-2
Environmental options, 8.6-8
Errors, 7.4-1
ETC interface tab, 5.2-6
ETCLink, 11.4-1
ETCLink functions, 11.4-1
Ethernet, 10.3-1
Exit WYSIWYG, 5.1-6
Export to DWG/DXF, 5.1-1
Extrude, 6.6-7

F

Facet resolution, 5.2-9
Facet splitting, 5.2-9
File menu, 5.1-1
Fillet, 6.6-7
Find tool, 7.2-3
Fixture layout, 6.8-16
Fixture notation, 6.8-16
Fixture snap, 6.2-7
Fixtures, 6.8-1
Flight Case, 6.10-1
Floor mount, 6.7-1, 6.7-6
Flow control, 5.2-2
Focus, 8.2-3
Focus data field, 7.2-1
Focus position, 6.8-19
Focus tool, 8.2-3
Footnotes data field, 7.2-2
French curve, 6.4-4
Front view, 6.2-5
Full screen mode, 4.1-4

G

Gel, 8.2-3
General object properties, 6.5-4
Gobo, 6.9-4
Gobo data field, 7.2-1
Gobo wheel, 8.2-3
Grid options, 5.2-13
Gridlines, 9.3-2
Grouping, 6.5-1

H

Hang structure, 6.8-1, 6.8-3
Hanging, 6.7-6, 6.8-1
Hanging fixtures, 6.8-1
Hardware acceleration, 5.2-3
Hatch style, 6.8-19
Hidden line, 6.4-4
Hookup data field, 7.2-1
Hot keys, 5.2-15

I

Image items, 9.3-8
Images, 9.4-1, 11.3-1
Imperial units, 6.2-2
Importing DWG/DXF files, 6.2-8, 6.2-9
Insert (Pres) toolbar, 4.5-5
Inserting fixtures, 7.2-4
Inserting focus positions in data mode, 7.2-4
Inserting items into plots, 9.3-2
Inserting positions in data mode, 7.2-5
Installation, 3.1-1
Installing
  WYSIWYG, 3.1-2
  WYSIWYG Learn, 3.1-3
  WYSIWYG Network, 3.1-3
Intensity design tool, 8.2-2
Interactive mode, 5.2-12, 6.2-7
Isometric view, 6.2-5
Item properties, 9.3-4

J

jpeg file, 6.9-5, 8.6-3, 9.3-2, 9.4-1

K

Keyboard and mouse control, 4.5-1

L

Label layout, 6.8-17
Ladders, 6.7-1, 6.7-6
Lamp Type data field, 7.2-2
Lamps, 6.3-2
Latitude, 5.2-11
Layer data field, 7.2-2
Layers, 6.5-13
Layers toolbar, 4.5-5
Layout
  fixture data, 6.8-16
Layout tabs
  CAD, 6.1-1
  Data, 7.1-1
  Design, 8.1-1
  Link, 11.1-1
  Live, 10.1-1
  Pres, 9.1-1

Left view, 6.2-4
Library Browser, 6.3-1
Library item, 6.5-10
Library shortcuts, 4.2-1
Light emission, 6.5-5
Line, 6.4-4
Live toolbar, 4.5-5
Load recording, 11.4-7
Longitude, 5.2-11

M

Masking, 6.4-9
Materials, 6.5-5, 6.5-10, 6.5-11
Measurement mode, 6.4-7
Menu bar, 4.5-1
Merge, 5.1-3
Message Log, 11.3-1
Message Log Options toolbar, 4.5-6
Metric units, 6.2-2
MIDI, 10.3-1
Mirror tool, 6.5-3
Mirroring, 6.5-3
Missing coordinate, 6.2-3
Model data field, 7.2-2
Modes, 4.1-1
Modifying wireframe views, 4.5-2
Move tool, 6.5-2
Moving, 6.5-2
Moving head, 6.8-14
Moving mirror, 6.8-14

N

Navigation shortcuts, 4.2-1
New Show, 5.1-1
Node Manager, 11.4-3
Number of colour frames data field, 7.2-2
Number of data channels data field, 7.2-2
Number of lamps data field, 7.2-2

O

Object-specific properties, 6.5-6
Offset, 6.7-5
Offset data field, 7.2-1
Open a show document, 5.1-1
Open GL tab, 5.2-4
Opening a new window, 4.1-4
Options, 5.2-1
Origin, 5.2-13, 6.2-1
Ortho, 4.4-1, 6.2-5
Owner data field, 7.2-2

P

Palette shortcuts, 8.4-1
Pan data field, 7.2-2
Passwords, 3.5-1
Patch, 7.3-1
Patch data field, 7.2-1
Patch universe, 7.3-1
Patching fixtures, 7.3-2
Photo quality pictures, 8.6-1
Photometrics, 6.8-1
Pipe, 6.4-9, 6.8-2
Pipe snap, 6.2-7
Pipe tape, 9.6-1
Placing, 6.9-1
Plan view, 6.2-4, 6.2-5
Plot items, 9.3-4
Plot type, 6.2-4
Plots (Pres mode), 9.3-1
Points, 6.4-5
Popup frames, 4.5-1
Position, 6.7-1
Position data field, 7.2-1
Position Manager, 6.7-1
POV (Point of View), 6.8-13
Precision, 5.2-14, 6.5-9
Presence option, 8.6-5
Presentation items, 9.3-2
Print, 5.1-4
Processors, 5.2-5
Properties
  Dimension, 6.5-9
  Errors, 7.4-3
  Focus position, 6.8-5
  General, 6.5-4
  Library item, 6.3-3, 6.5-10
  Line, 6.5-7
  Patch universe, 7.3-3
  Pipe, 6.7-5
  Plot, 9.3-1
  Point, 6.5-7
  Report, 9.2-1
  Spreadsheet, 7.2-6
  Symbol, 6.8-19
  Text label, 6.5-8
  Wireframe view, 4.5-2
Proscenium Arch, 6.4-1
Purpose data field, 7.2-1

Q

Quick focus, 6.8-5
Quick tools, 6.9-6, 6.9-7

R

Radiosity, 8.6-7
Reading the patch, 7.3-2
Redesigning, 9.2-2
Redesigning headings, 9.2-2
Regional profile, 5.2-2
Regional settings, 5.2-10
Registration, 3.2-1
Relative, 6.5-2
Relative coordinates, 4.4-2
Render options, 5.2-5

Render shortcuts, 8.6-1
Rendering, 6.11-1, 8.6-1, 10.6-1
Replace tool, 7.2-3
Replacing fixtures, 6.8-1
Report headings, 9.2-2
Reports, 9.2-1
Right view, 6.2-4
Riser, 6.4-7
Rotate tool, 6.5-3
Rotating, 6.5-3

S

Saturation, 8.6-9
Save, 5.1-2
Save As, 5.1-2
Saving a lighting look, 8.5-1
Scenes, 6.5-15
Scenes toolbar, 4.5-6
Scrollbars, 4.5-4
Scrollers, 6.9-4, 8.2-3
Selecting objects, 6.5-1
Send the show document through email, 5.1-6
Set Piece tab, 6.5-6
Setting display size for shortcuts, 4.2-3
Settings, 5.2-1
  regional, 5.2-10
Shaded views, 4.6-1
Shortcut, 4.2-1
Shortcut bar
  Create Shortcuts
    Concepts, 8.3-1
    Images, 9.4-1
    Library Objects, 6.3-3
    Lighting looks, 8.5-1
    Palette, 8.4-1
    Patch Universes, 7.3-1
    Plots, 9.3-1
    Render, 8.6-1
    Reports, 9.2-2
    Spreadsheets, 7.2-7
    Worksheets, 9.5-1
Shortcuts, 4.2-1
  arranging, 4.2-3
  delete, 4.2-2
  new, 4.2-2
  ordering, 4.2-2
  setting display size, 4.2-3
Show, 5.2-7
Show details, 5.2-14
Show information, 9.2-4
Show Options, 5.2-7
Shutter cuts, 6.8-6
Side view, 6.2-5
Simulation options, 5.2-14, 6.11-1, 10.6-1
Snap, 4.4-1, 5.2-13, 6.2-6
Soft goods, 6.4-9
Soft shadow, 8.6-8
Solid line, 6.4-4
Sphere, 6.4-8
Spin data field, 7.2-2
Spline, 6.4-4, 6.4-5
Spot, 6.5-5
Spot data field, 7.2-1
Spreadsheets, 7.2-1
Standard toolbar, 4.5-6
Status bar, 4.4-1
Status data field, 7.2-2
Stock, 6.9-2
Subscriptions, 3.4-1
Surfaces, 6.4-7
Symbols, 5.2-9, 5.2-14, 6.8-16

T

Tag data field, 7.2-2
Template, 5.1-3
Text label, 6.4-6
Textures, 5.2-3, 6.5-10
Tilt data field, 7.2-2
Time zones, 5.2-11
Title block, 9.2-4, 9.3-2
Toolbars, 4.5-1, 5.2-4
Tools
  CAD, 6.6-1
  Design, 8.2-1
Tools toolbar, 4.5-7
Tooltips, 5.2-14
Tophats, 6.9-1
Troubleshooting, 1.2-1
Truss, 6.7-6
Truss cross members, 5.2-14
Type data field, 7.2-1

U

Ungrouping, 6.5-1
Unit data field, 7.2-1
Universes toolbar, 4.5-7
Unused channels, 4.4-1
Upgrades, 3.3-1
USB serial converter, 5.2-2
User Options, 5.2-11
USITT symbols, 6.8-19

V

Venue, 6.4-1
Vertex, 6.6-1, 6.6-6
Vertical justification of a line, 6.5-9
View - full screen, 4.1-4
View Options
  Error, 7.4-3
  Flight Case, 6.10-3
  Patch universe, 7.3-3
  Plot, 9.3-1
  Report, 9.2-1
  Shaded views, 4.6-2
  Spreadsheet, 7.2-6
  Wireframe views, 4.5-2
View shortcuts, 4.5-1

View toolbar, 4.5-8
Views/tools shortcuts, 4.2-2
Visualization, 10.7-1

W

Wattage data field, 7.2-2
Weight data field, 7.2-2
Welcome window, 4.1-1
Wheels, 6.9-4, 8.2-3
Wireframe views, 4.5-1
Workplane, 6.2-5
WYSILink, 11.2-1
WYSILink toolbar, 4.5-8, 11.4-1
WYSIWYG, 2.4-1, 2.6-1
  Design, 2.2-1
  Perform, 2.3-1, 2.5-1, 2.7-1
  Report, 2.1-1
WYSIWYG Learn, 2.5-1, 3.1-3
WYSIWYG Network, 2.6-1
WYSIWYG Perform - Console Edition, 1.2-2, 2.4-1
WYSIWYG Viewer, 5.3-1

Z

Zoom, 8.2-1